C O N T E N T S

MSC.Patran PAMCRASH Preference Guide

CHAPTER 1
Overview

- Purpose, 2
- PAMCRASH Product Information, 3
- What is Included with this Product?, 3
- PAMCRASH Preference Integration with MSC.Patran, 4
- MSC.Patran PAMCRASH Preference Components, 5

2
Building A Model

- Introduction to Building a Model, 8
- Coordinate Frames, 12
- Finite Elements, 13
  - Nodes, 14
  - Elements, 15
- Material Library, 16
  - Materials Form, 17
- Element Properties, 27
  - Element Properties Form, 28
    - 0D Mass, 30
    - Beam, 31
    - Rod, 32
    - Linear Spring, 33
    - Shell, 34
    - Solid, 36
- Loads and Boundary Conditions, 37
  - Loads & Boundary Conditions Form, 38
    - Time Dependent (Time Varying), 41
    - Contact Toolkit, 42
    - Object Tables, 47
- Load Cases, 55

3
Running an Analysis

- Review of the Analysis Form, 58
  - Analysis Form, 59
- Translation Parameters, 61

- Solution Parameters, 62
  - Solution Control, 63
  - Global Damping, 64
  - Solid Viscosity, 64
  - Shell Control, 65
- Select Load Case, 66
- Output Requests, 67
- Output Controls, 69
- Select Group, 70
- Setting PAMCRASH IDs, 71

4
Read Input File

- Review of Read Input File Form, 74
  - Read Input File Form, 75
- Selection of Input File, 76
- Data Translated from the PAMCRASH Input File, 77
- Reject File, 79

5
Files

- Files, 82

INDEX

- MSC.Patran PAMCRASH Preference Guide, 83

CHAPTER
1

Overview

- Purpose
- PAMCRASH Product Information
- What is Included with this Product?
- PAMCRASH Preference Integration with MSC.Patran
- MSC.Patran PAMCRASH Preference Components

1.1 Purpose

MSC.Patran is an analysis software system developed and maintained by MSC.Software Corporation. The core of the system is MSC.Patran, a finite element analysis pre- and post-processor. The MSC.Patran system also includes several optional products such as advanced postprocessing programs, tightly coupled solvers, and interfaces to third party solvers. This document describes one of these interfaces.

The MSC.Patran PAMCRASH Application Preference provides a communication link between MSC.Patran and PAMCRASH. It also provides customization of certain features that can be activated by selecting PAMCRASH as the analysis code "Preference" in MSC.Patran.

The PAMCRASH Preference is fully integrated into MSC.Patran. The casual user will never need to be aware separate programs are being used. For the expert user, there are two main components of the preference: a *PCL* function, `load_pamcrash()`, which will load all PAMCRASH specific definitions, like element types and material models, into the currently opened database, and the `pat3pam` *program used* to convert model data from the MSC.Patran database into the analysis code input file, and to translate model topology from the analysis code input file into the MSC.Patran database.

Selecting PAMCRASH as the analysis code under the "Analysis Preference" menu modifies MSC.Patran forms in *five* main areas:

1. Materials
2. Element Properties
3. Finite Elements / MPCs and Meshing
4. Loads and Boundary Conditions
5. Analysis forms

The *PCL* function `load_pamcrash()` can be invoked by simply typing its name into the MSC.Patran command line. This will load PAMCRASH specific definitions into the MSC.Patran database currently opened. PAMCRASH specific definitions can be added to any MSC.Patran database (which does not already contain PAMCRASH specific definitions) at any time. Obviously, a MSC.Patran database must be open for `load_pamcrash()` to operate correctly. See **PAMCRASH Preference Integration with MSC.Patran** (p. 4) for complete information and a description of how to create a new template database.

The `pat3pam` *program* translates model data between the MSC.Patran database and the analysis code-specific input file format. This translation must have direct access to the originating MSC.Patran database when PAMCRASH input file is being created.

The `pat3pam` program also translates model topology data from the analysis, code-specific input file into the MSC.Patran database. When reading an existing PAMCRASH input file the MSC.Patran database must be initially empty.

Reading PAMCRASH Input Files. This release of the MSC.Patran PAMCRASH Preference provides support for reading PAMCRASH input files. Nodes, elements, materials, LBCs, Property Sets, and coordinate systems from keyword based input files only are supported.

Post Processing PAMCRASH Results. No Postprocessing of PAMCRASH analysis results is currently available in MSC.Patran. It is recommended to use PAMCRASH postprocessor PAMVIEW for this purpose.

1.2 PAMCRASH Product Information

PAMCRASH is a general purpose explicit finite element computer program for nonlinear dynamic analysis of structures in three dimensions. PAMCRASH is integrated into the PAMSOLID library of solvers.

The program is developed, supported and maintained by Pam System International, ESI Group Software Product Company, 20, Rue Saarinen, Silic 303, 94588 Rungis Cedex. 33 (1) 49 78 2800. See the PAMCRASH / PAMSAFE User's Manual for a general description of the software's capabilities.

1.3 What is Included with this Product?

The MSC.Patran PAMCRASH Preference product includes the following items:

1. A PCL function contained in `p3patran.plb` which will add PAMCRASH specific definitions to any MSC.Patran database (not already containing such definitions) at any time.
2. A PCL library called `pamcrash.plb` and contained in the <installation_directory> directory. This library is used by the analysis forms to produce analysis code specific translation parameter, solution parameter, etc. forms.
3. An executable program called `pat3pam` contained in the <installation_directory> / bin / exe directory. This program translates information from PAMCRASH input files into an MSC.Patran database and translate information from an MSC.Patran database into a PAMCRASH input file. The program can be run independently of MSC.Patran but is typically run underneath MSC.Patran, transparent to the user.
4. This MSC.Patran PAMCRASH Preference Guide is included as part of the product. An online version is also provided to allow direct access to this information from within MSC.Patran.

1.4 PAMCRASH Preference Integration with MSC.Patran

Creation of a PAMCRASH Template Database. Two versions of the MSC.Patran database are delivered with MSC.Patran. Both occur in the <installation_directory> directory and they are named `base.db` and `template.db`. The `base.db` database is a MSC.Patran database into which no analysis code specific definitions, such as element types and material models, have been stored. The `template.db` database is a version of the MSC.Patran database which contains analysis code specific definition needed by a number of the MSC supplied interfaces. In order to create a template database which contains only PAMCRASH specific definitions, the user should follow these steps:

1. Within MSC.Patran open a new database using `base.db` as the template.
2. Enter `load_pamcrash()` into the command line.
3. Save this database under a name like `pamcrash.db` to be your new "PAMCRASH only" template database.
4. From then on, when opening a new database, choose `pamcrash.db` as your template database.

Any databases derived from `base.db` may not contain the needed PAMCRASH specific definitions needed to run the PAMCRASH Preference. But, PAMCRASH specific definitions can be added to any database at any time by simply typing `load_pamcrash()` into the MSC.Patran command line while the target database is the database currently opened by MSC.Patran. Due to the savings in size and for the sake of simplicity it is highly recommended `template.db` not be used as a template database and that the user create their own unique template database which contains only the analysis code specific definitions pertaining to the analysis codes of immediate interest. For more details about adding analysis code specific definitions to a database and / or creating unique template databases, refer to **Modifying the Database Using PCL** (p. 347) in the *PCL and Customization* or to the ***MSC.Patran Installation and Operations Guide***.

1.5 MSC.Patran PAMCRASH Preference Components

The diagrams shown below indicate how the functions, scripts, programs, and files which constitute the PAMCRASH Preference affect the MSC.Patran environment. Site customization, in some cases, is indicated.

Figure 1-1 shows the process of running an analysis. The `pamcrash.plb` library defines the Translation Parameter, Solution Type, Solution Parameter, and Output Request forms called by the Analysis form. When the *Apply button* is pushed on the Analysis form `pat3pam` is executed. `pat3pam` reads data from the database and creates the PAMCRASH input file. If `pat3pam` finishes successfully, and the user requests it, the script will then start PAMCRASH.

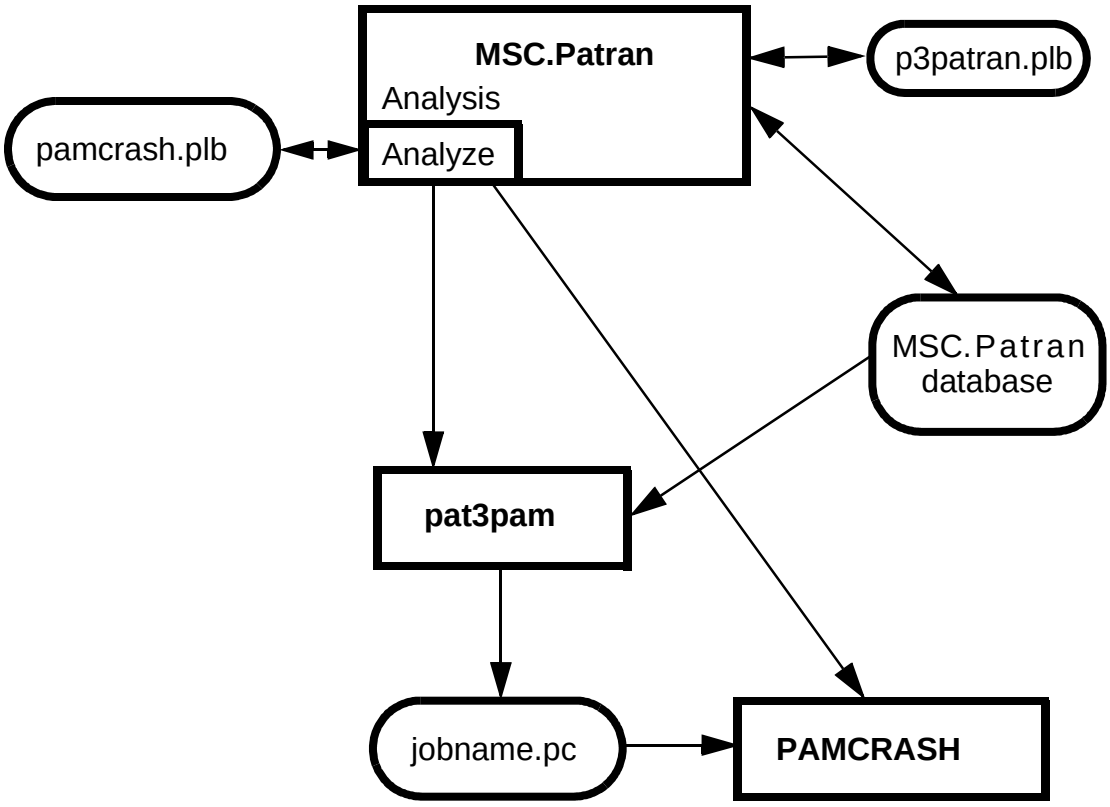


Figure 1-1 Forward Translation

Figure 1-2 shows the process of translating information from a PAMCRASH input file into a MSC.Patran database. The behavior of the main Analysis/Read input file form and the subordinate Select input file form is dictated by the `pamcrash.plb` PCL library. The apply button on the main form activates the `pat3pam` program which reads the specified PAMCRASH input file into the MSC.Patran database.

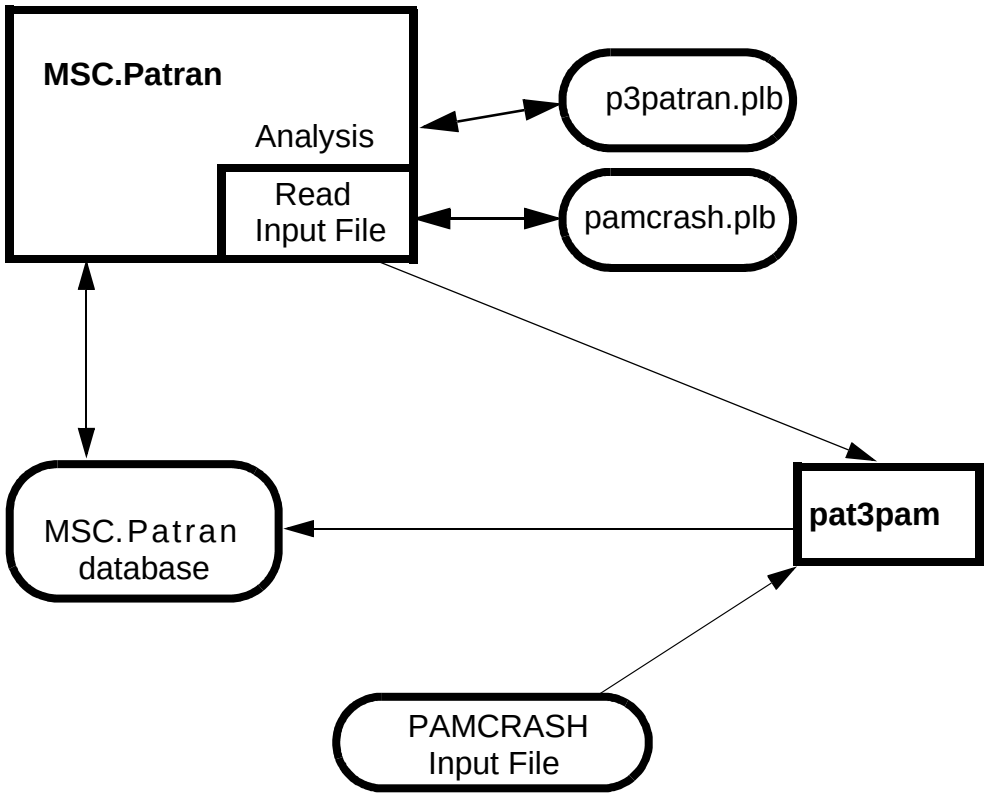


Figure 1-2 PAMCRASH Input File Translation

CHAPTER

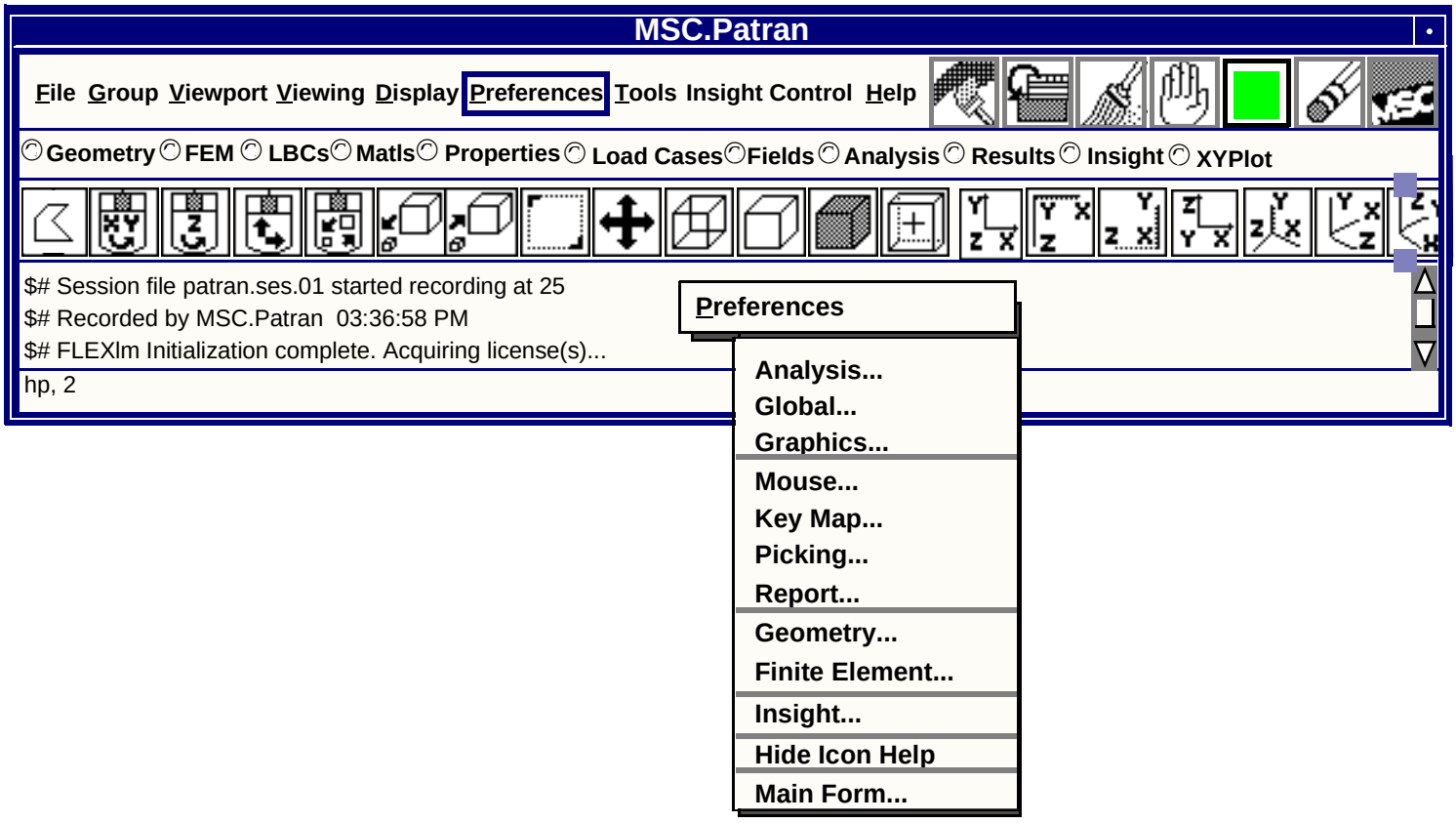
2 Building A Model

- Introduction to Building a Model
- Coordinate Frames
- Finite Elements
- Material Library
- Element Properties
- Loads and Boundary Conditions
- Load Cases

2.1 Introduction to Building a Model

There are many aspects to building a finite element analysis model. In several cases, the forms used to create the finite element data are dependent on the selected analysis code and analysis type. Other parts of the model are created using standard forms.

Under Preferences on the MSC.Patran main form is a selection that defines the intended analysis code to be used for this model.



The analysis code may be changed at any time during model creation.This is especially useful if the model is to be used for different analyses, in different analysis codes. As much data as possible will be converted if the analysis code is changed after the modeling process has begun. The analysis option defines what will be presented to the user in several areas during the subsequent modeling steps.

These areas include the material and element libraries, including multi-point constraints, the applicable loads and boundary conditions, and the analysis forms. The selected Analysis Type may also affect the allowable selections in these same areas. For more details, see **The Analysis Form** (p. 8) in the *MSC.Patran Reference Manual, Part 5: Analysis Application*.

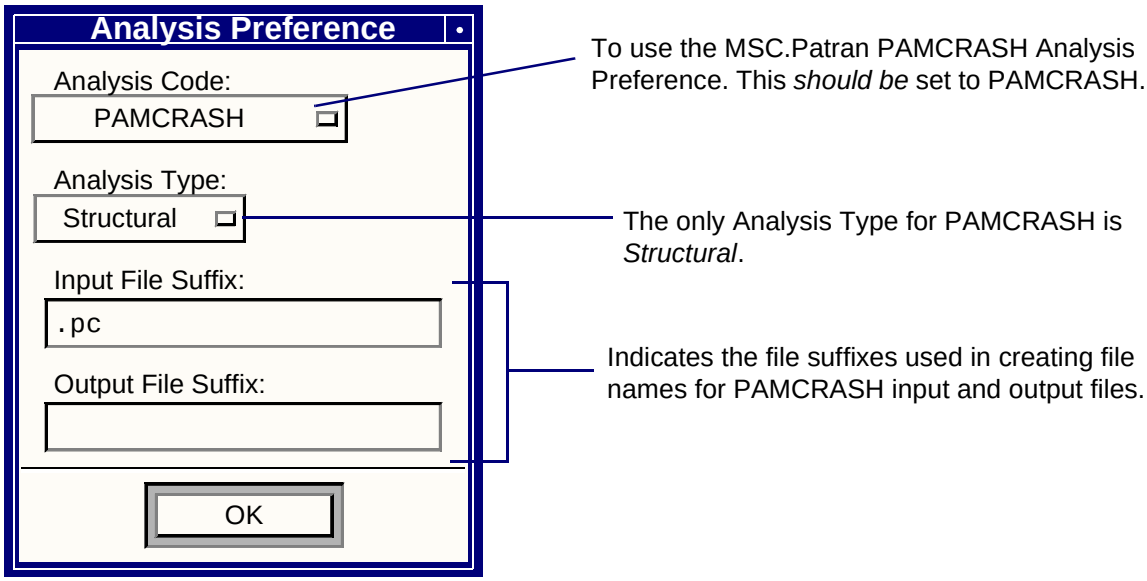

Analysis Preference
Analysis Code:
PAMCRASH
Analysis Type:
Structural
Input File Suffix:
.pc
Output File Suffix:
OK
To use the MSC.Patran PAMCRASH Analysis Preference. This *should be* set to PAMCRASH.
The only Analysis Type for PAMCRASH is *Structural*.
Indicates the file suffixes used in creating file names for PAMCRASH input and output files.

Table 2-1 summarizes the various PAMCRASH commands supported by the MSC.Patran PAMCRASH Preference.

Table 2-1 Supported PAMCRASH Entities

File Section	Keyword	Method
CONTROL	FREE (Free Format)	
	NOLIST / LIST (Listing Control)	Analysis / Output Controls
	NOPRINT / PRINT (Printing Control)	Analysis / Output Controls
	MNTR (Monitoring)	Analysis / Output Controls
	FILE (File Name)	Analysis
	CPULIMIT (CPU Limit)	Analysis / Solution Parameters / Solution Control
	SHELLCHECK (Shell Geometry Limits)	Analysis / Solution Parameters / Shell Control
	DATACHECK (Data Checking)	Analysis / Solution Parameters / Solution Control
	TIMESTEP (Shell Time Step Control)	Analysis / Solution Parameters / Shell Control
	TITLE_ / _ (Job Title)	Analysis
	INCLU_ / _ (Include File)	Analysis / Translation Control
	CTRL__ / _	Analysis /

Table 2-1 Supported PAMCRASH Entities (continued)

File Section	Keyword	Method
MATERIAL	SOLID TYPE 1 (Elastic Plastic, bilinear and stress / strain)	Materials / Properties (3D)
	SOLID TYPE 2 (Crushable Foam)	Materials / Properties (3D)
	SOLID TYPE 5 (Viscoelastic)	Materials / Properties (3D)
	SOLID TYPE 99 (Null)	Materials / Properties(3D)
	SHELL TYPE 100 (Null)	Materials / Properties(2D)
	SHELL TYPE 101 (Elastic)	Materials / Properties(2D)
	SHELL TYPE 102 / 103 (Elastic Plastic)	Materials / Properties(2D)
	SHELL TYPE 105 / 106 (Elastic Plastic with damage)	Materials / Properties(2D)
	SHELL TYPE 130 (composite)	Materials / Properties(2D)
	BEAM / BAR TYPE 200 (Null)	Materials / Properties (1D)
	BEAM / BAR TYPE 201 (Elastic)	Materials / Properties (1D)
	BEAM / BAR TYPE 202 (Elastic Plastic)	Materials / Properties (1D)
	BAR / DASHPOT TYPE 204 (Nonlinear)*	Materials / Properties (1D)
	BEAM TYPE 212 (Elastic Plastic)	Materials / Properties (1D)
PLY DATA	PLY_ _ _/ _(Composite)	Materials / Properties(2D)
NODES	FRAME_/ _ (Coordinate Frame)	Geometry
	NODE__/ _ (Nodal Point Data)	Finite Elements
	MASS__/ _ (Added Mass)	Properties (0D)
	BOUNC_/ _ (Displacement Boundary Condition)	LBC's
	INVEL_/ _ (Initial Velocity)	LBC's
	VELBC_/ _ (Velocity Boundary Condition)	LBC's
	CONLO_/ _ (Concentrated Loads & Follower Forces)*	LBC's
	DAMP__/ _ (Nodal Damping by Group)	LBC's
ELEMENTS	SOLID_/ _ (Solid Elements)	Finite Elements
	SHELL_/ _ (Shell Elements)	Finite Elements
	BEAM__/ _ (Beam Elements)	Finite Elements
	BAR___/ _ (Bar Elements)	Finite Elements
	SPRING / _ (Spring Elements)	Finite Elements

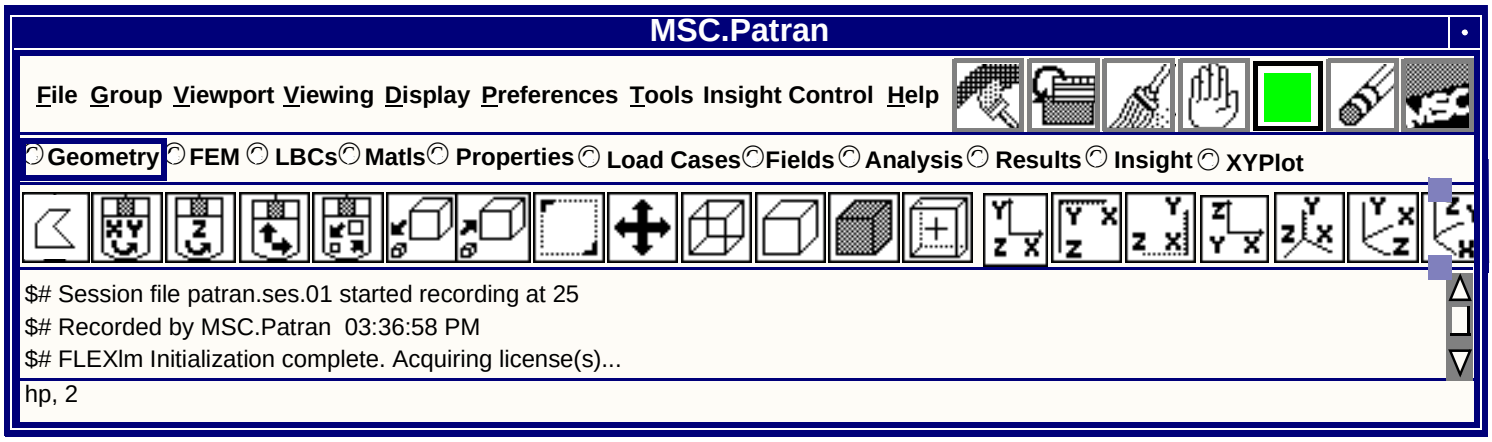
Table 2-1 Supported PAMCRASH Entities (continued)

File Section	Keyword	Method
CONSTRAINT	RIGWA_/_ (Rigid Walls)	LBC's
	NODCO_/_ (Nodal Constraints)	LBC's
	RIGBO_/_ Rigid Body (Regular only)	LBC's
	SLINT2_/_ (Sliding Interfaces)	LBC's
AUXILIARY	FUNCT_/_ (Function)	LBC's
PLOT OUTPUT	THLNO_/_ (Nodal Time History)	Analysis / Output Requests
	THLOC_/_ (Local Coordinate System)	Analysis / Output Requests
	THLSO_/_ (Solid Element Output)	Analysis / Output Requests
	THLSH_/_ (Shell and Membrane Element Output)	Analysis / Output Requests
	THLBM_/_ (Beam, Bar, Spring / Dashpot etc. Output)	Analysis / Output Requests
	TRAFO_/_(Cross sections for Force output)	Analysis / Output Requests
	SECFO_/_(Cross sections for Force output)	Analysis / Output Requests

* Note that Non-Linear Springs and Follower Forces are not supported in this version of the MSC.Patran PAMCRASH Preference.

2.2 Coordinate Frames

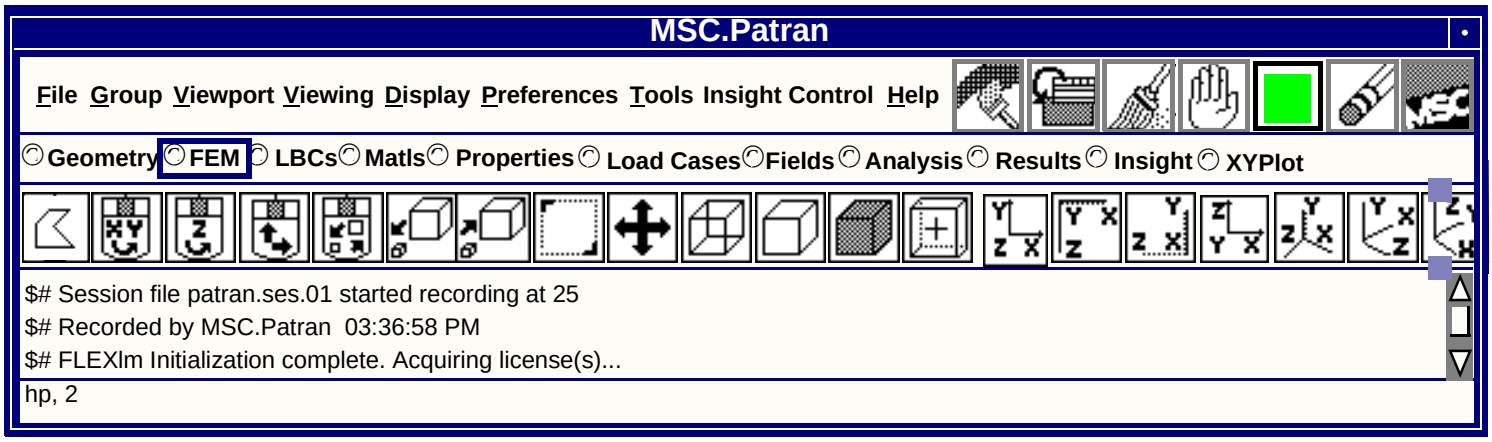
Coordinate frames will generate unique **FRAME_/_** entries.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Only Coordinate Frames which are referenced by nodes, element properties, or loads and boundary conditions can be translated. Note that Coordinate Frames used to define skewed boundary conditions in MSC.Patran will be translated even though they are not required in PAMCRASH. For more information on creating coordinate frames see **Creating Coordinate Frames** (p. 350) in the *MSC.Patran Reference Manual, Part 2: Geometry Modeling*.

2.3 Finite Elements

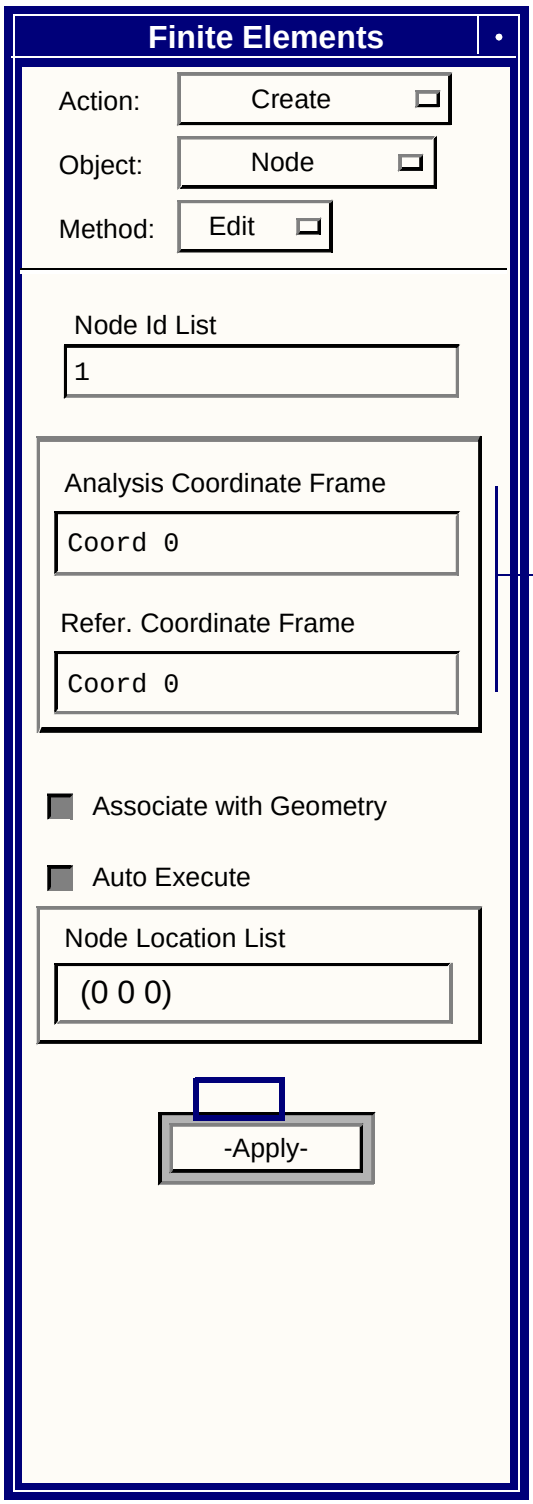
Finite Elements in MSC.Patran allows the definition of basic finite element construction. Created under Finite Elements are the *nodes and element topology*.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

For more information on how to create finite element meshes, see **Mesh Seed and Mesh Forms** (p. 29) in the *MSC.Patran Reference Manual, Part 3: Finite Element Modeling.*

Nodes

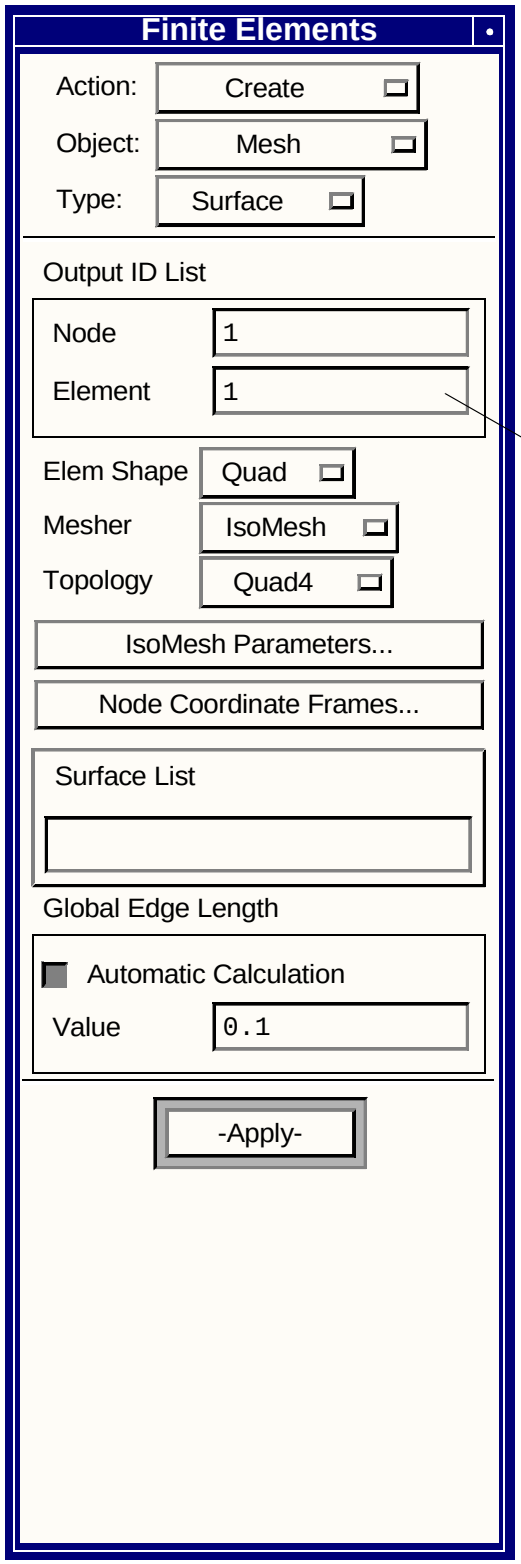
Nodes in MSC.Patran will generate unique **NODE__/_** data entries. Nodes can be created either directly using the *Node object*, or indirectly using the *Mesh object*. Note that unconnected Nodes may be used in a PAMCRASH model to define beam orientations and the optional centre of gravity nodes of rigid bodies. These nodes should not be deleted.



The analysis frame applies for the entire model. The reference coordinate system is used during node generation only.

Elements

Finite Elements in MSC.Patran assigns element connectivity, such as Quad / 4, for standard finite elements. The type of PAMCRASH element to be created is not determined until the element properties are assigned. See the **Element Properties Form** (p. 28) for details concerning the PAMCRASH element types. Elements can be created either discretely using the Element object or indirectly using the Mesh object.



Elements which are not referenced by an element property region which is understood by the MSC.Patran PAMCRASH forward translator will not be translated.

Note: Previous versions of the MSC.Patran PAMCRASH Preference would translate tri-elements as degenerate quads in the analysis file “Which would be read back into the MSC.Patran database as tris”. This version of the preference translates tris as tris in both directions of translation. Similarly degenerate quads are translated as degenerate quads in both directions of translation.

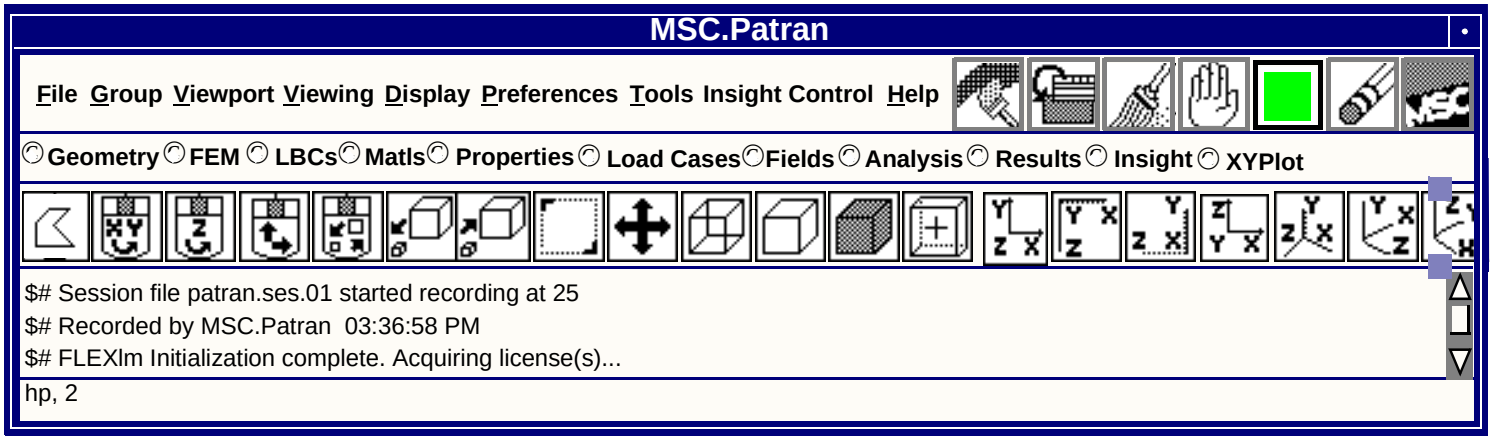
2.4 Material Library

The Materials form will appear when the Materials toggle, located on the MSC.Patran application selections, is chosen. The selections made on this form will determine which Materials form appears and, ultimately, which PAMCRASH material will be created.

Several materials within PAMCRASH differ only by the material ID, even though the material models are identical. The difference in material ID is due to the underlying element dimensionality. e.g. PAMCRASH Materials 1, 102 and 212 are all elastic / plastic material models which share the same input data, however Material Type 1 is applicable only to solids, Type 102 only to shells and Type 212 only to beams. Within MSC.Patran, all of these materials are defined as a single material model which may be applied to any of the applicable element types. When the translator is called to produce a PAMCRASH input file, appropriate PAMCRASH material types are created for each element type required.

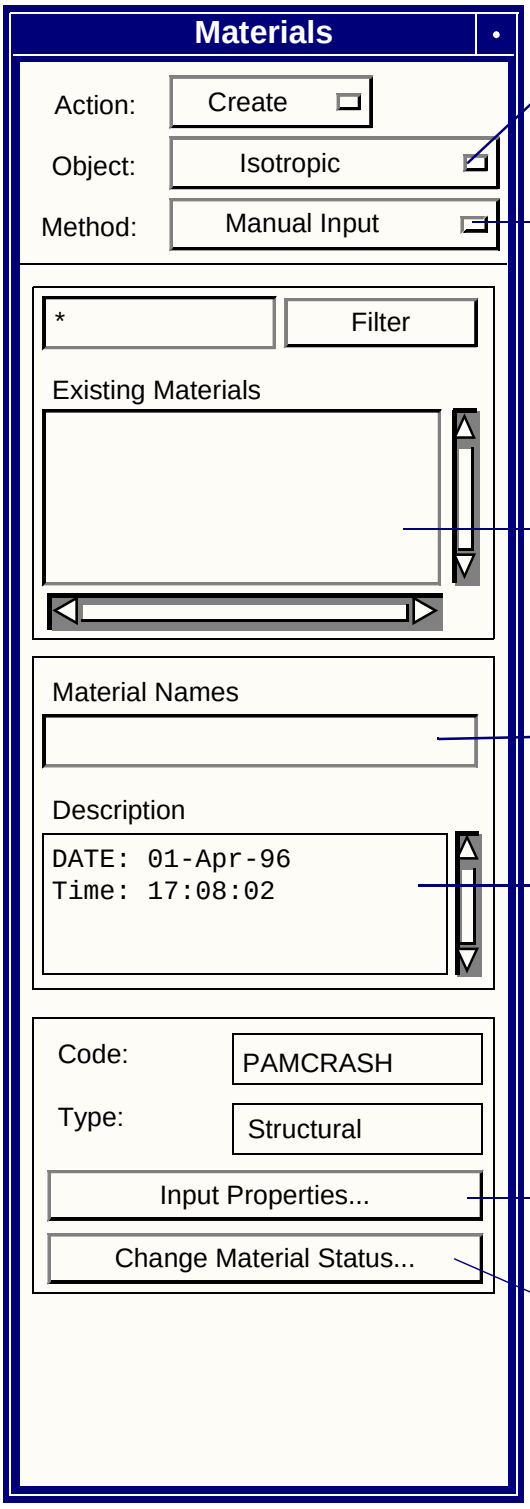
The following pages give an introduction to the Materials form, and details of all the material property definitions supported by the MSC.Patran PAMCRASH Preference.

Only material records which are referenced by an element property region or by a laminate lay-up will be translated. References to externally defined materials will result in special comments in the PAMCRASH input file, with material data copied from user identified files. This allows a user not only to insert material types that are not supported directly by the PAMCRASH preference, but also to make use of a standard library of materials.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Materials Form

This form appears when Materials is selected on the main form. The Materials form is used to provide options to create the various PAMCRASH materials.



This toggle defines the basic material orthotropy, and can be set to Isotropic or Composite.

The Method may be Manual Input, Materials Selector or Externally Defined. If “Externally Defined,” this form will have an Apply button which is used to ensure that the named material is added to the set of available materials.

Lists the created materials whose names pass the filter.

Defines the material name. A unique material ID will be assigned during translation.

Describes the material that is being created.

Indicates the active analysis code and analysis type. These selections are made by **Preferences>Analysis** (p. 343) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

Generates a form that is used to define the material properties.

Generates a form that is used to indicate the active portions of the material model. By default, all portions of a created material model are active.

Table 2-2 outlines the options when Create is the selected Action.

Table 2-2 PAMCRASH Materials

Object	Option 1	Option 2	Option 3
Isotropic	❑ Linear Elastic (1 / 101 / 201)		
	❑ Elastoplastic (1 / 102 / 103 / 105 / 106 / 202 / 212)	❑ Bilinear	Standard Iterative Isotropic Damage Anisotropic Damage
		❑ Single Curve	Standard Iterative Isotropic Damage Anisotropic Damage
	❑ Viscoelastic (5)		
	❑ Null / Rigid (99 / 100 / 200)		
	❑ Foam (2)		
Composite	❑ Laminate		

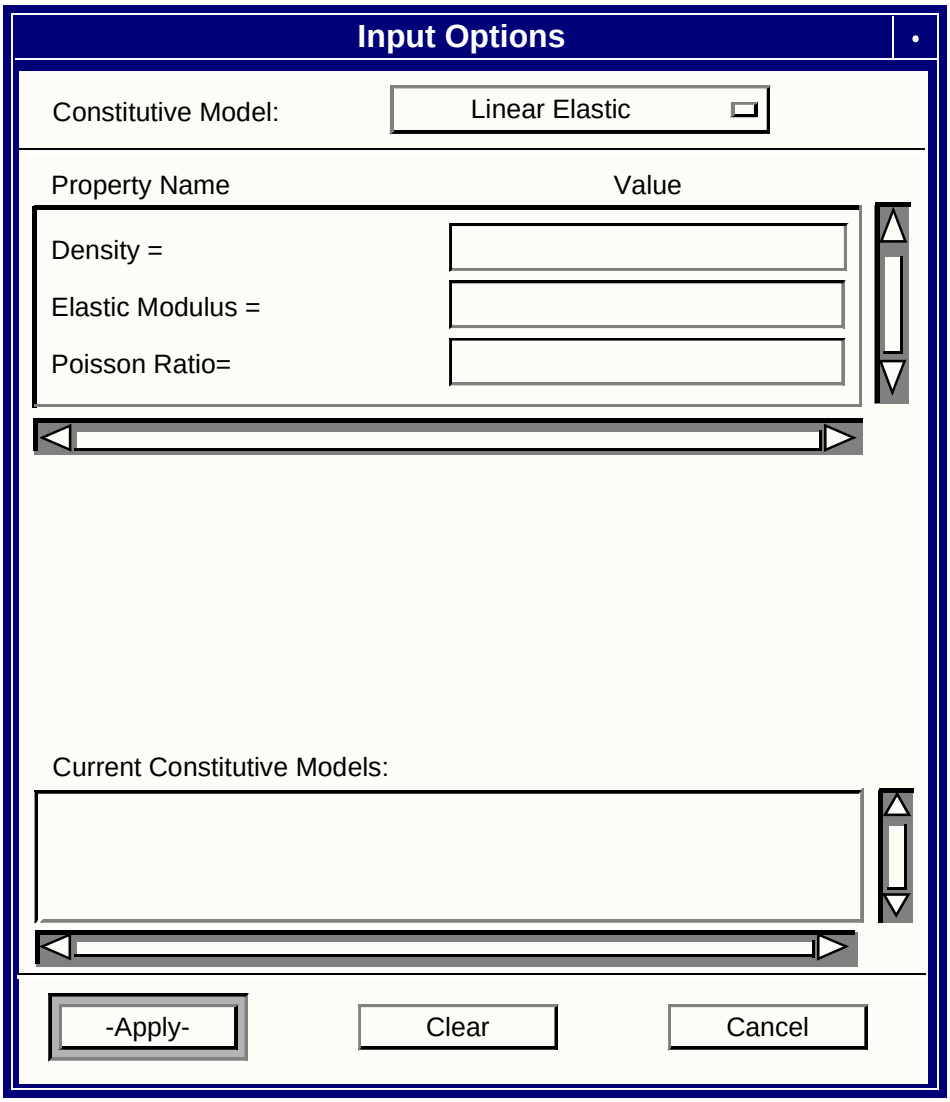
Isotropic

Linear Elastic

This subordinate form appears when the Input Properties button is selected on the Materials form when Isotropic is the object on the Material form, and when Linear Elastic is the selected Constitutive Model on the Input Options form.

Use this form to define the data for PAMCRASH Material Types 1, 101 and 201. The parameters required are: Density, Elastic Modulus and Poisson Ratio.

Note that PAMCRASH does not directly support a linear elastic material for solid elements. When this MSC.Patran material option is applied to solid elements, a PAMCRASH material Type 1 (Elastic / Plastic) will be written to the PAMCRASH input file, with the tangent modulus equal to the elastic modulus, and the Yield stress set artificially high (equal to the value of the elastic modulus).



Elastoplastic

The following subordinate forms appear when the Input Properties button is selected on the Materials form when any of the options are selected.

Table 2-3 Elastoplastic Material Options

Object	Option 1	Option 2	Option 3
Isotropic	Elastoplastic	Bilinear	Standard Iterative Isotropic Damage Anisotropic Damage
		Single Curve	Standard Iterative Isotropic Damage Anisotropic Damage

Use the forms on the next pages to define the data for PAMCRASH Material Types 1 (Solid), 102 / 103 / 105 / 106 (Shell), or 202 / 212 (Rod and Beam). Note that the strain rate dependency model is determined at a global level, by the ISTRAT parameter. Hence the 6 strain rate parameters cannot have names. For the sake of mapping to other codes the first two of those parameters should be the Cowper Symonds D and p parameters. The contents of the form will vary depending upon which option is selected. The parameters which are required are tabulated below.

Table 2-4 Elastoplastic Parameters

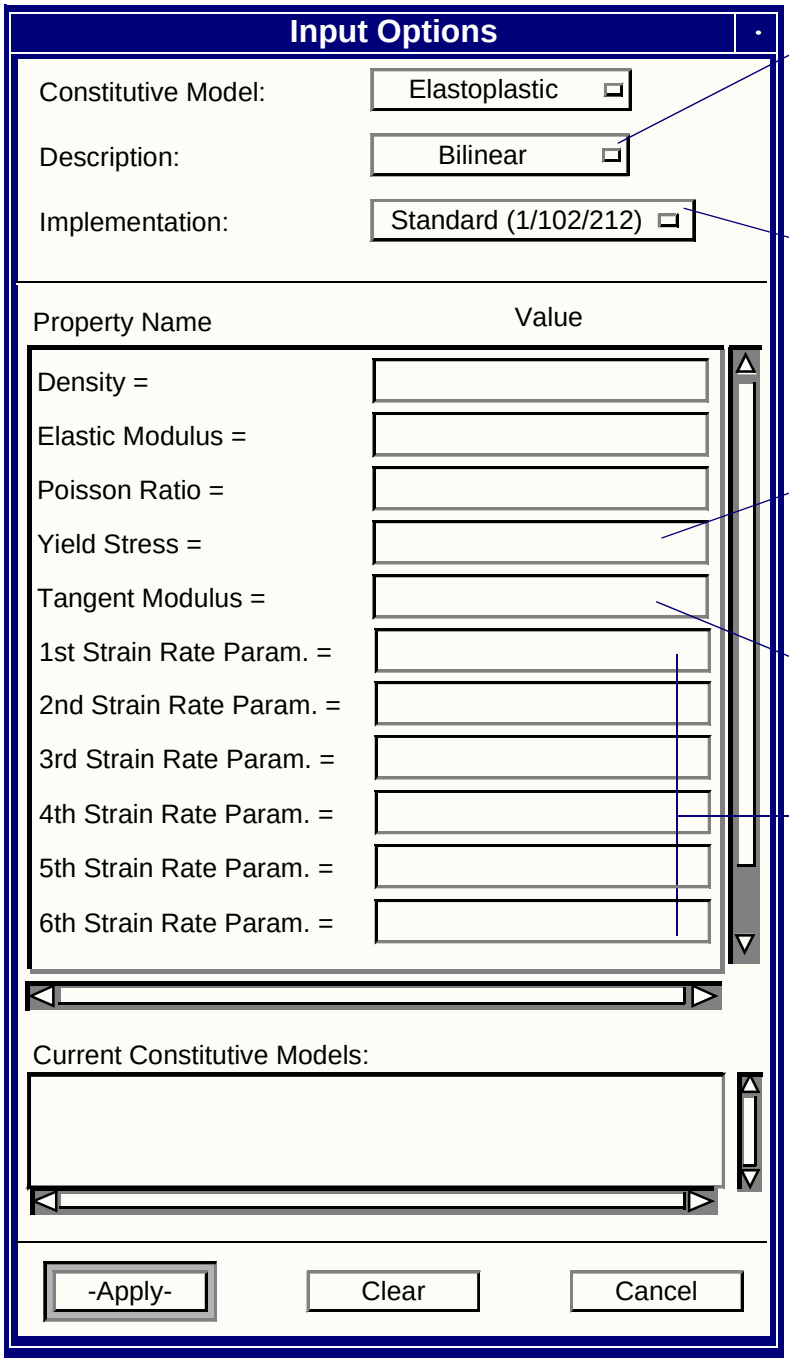
Parameter	Bilinear	Curve	Standard/Iterative	Iso/Anisotropic Damage
Density	x	x	x	x
Elastic Modulus	x	x	x	x
Poisson's Ratio	x	x	x	x
Yield Stress	x	Field	x	x
Tangent Modulus	x		x	x
1st Rate Param.	x	x	x	x
2nd Rate Param.	x	x	x	x
3rd Rate Param.	x	x	x	x
4th Rate Param.	x	x	x	x
5th Rate Param.	x	x	x	x
6th Rate Param.	x	x	x	x

Table 2-4 Elastoplastic Parameters

Parameter	Bilinear	Curve	Standard/Iterative	Iso/Anisotropic Damage
Initial Threshold				x
Inter. Threshold				x
Inter. Damage				x
Ultim. Threshold				x
Ultim Damage				x

Elastoplastic

The following form is typical of Elastoplastic material Input data forms when the Bilinear definition method is selected. Use this form to define the data for PAMCRASH Materials Types 1, 101, and 102.



The Bilinear option requires definition of the tangent modulus. For the “Single Curve” option the yield stress is replaced by a field defining effective stress vs effective plastic strain.

Choose between the Standard, Iterative, Isotropic Damage and Anisotropic Damage implementations.

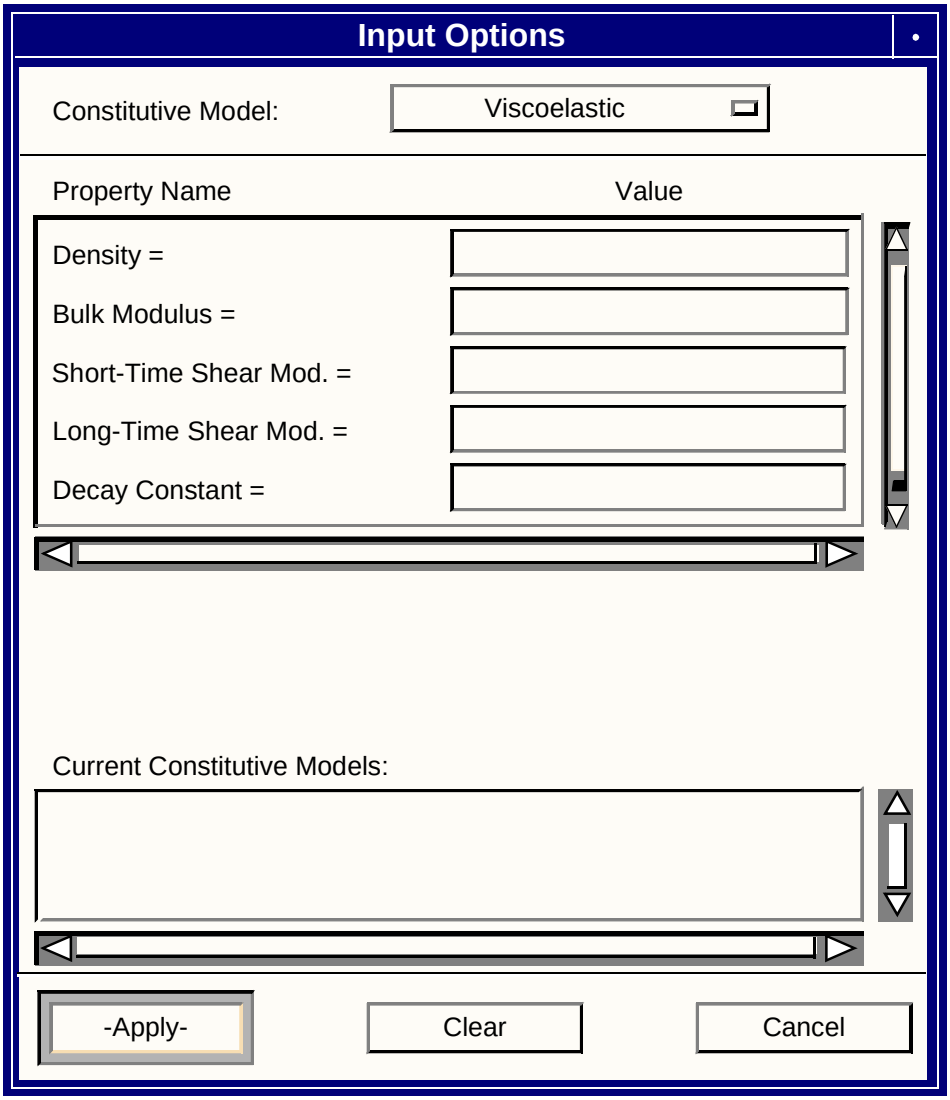
Note that this is a Strain Field if the Single Curve option is selected. Note, this translates to one curve in the “CURVE” definition rather than using the “Single Stress Strain” option.

The tangent modulus is only required for the bilinear curve type.

These parameters are required only if rate dependency is required. The first two terms are the Cowper Symonds D and p parameters if that model was selected on the Analysis Forms. Damage Parameters will be present depending on the selected implementation.

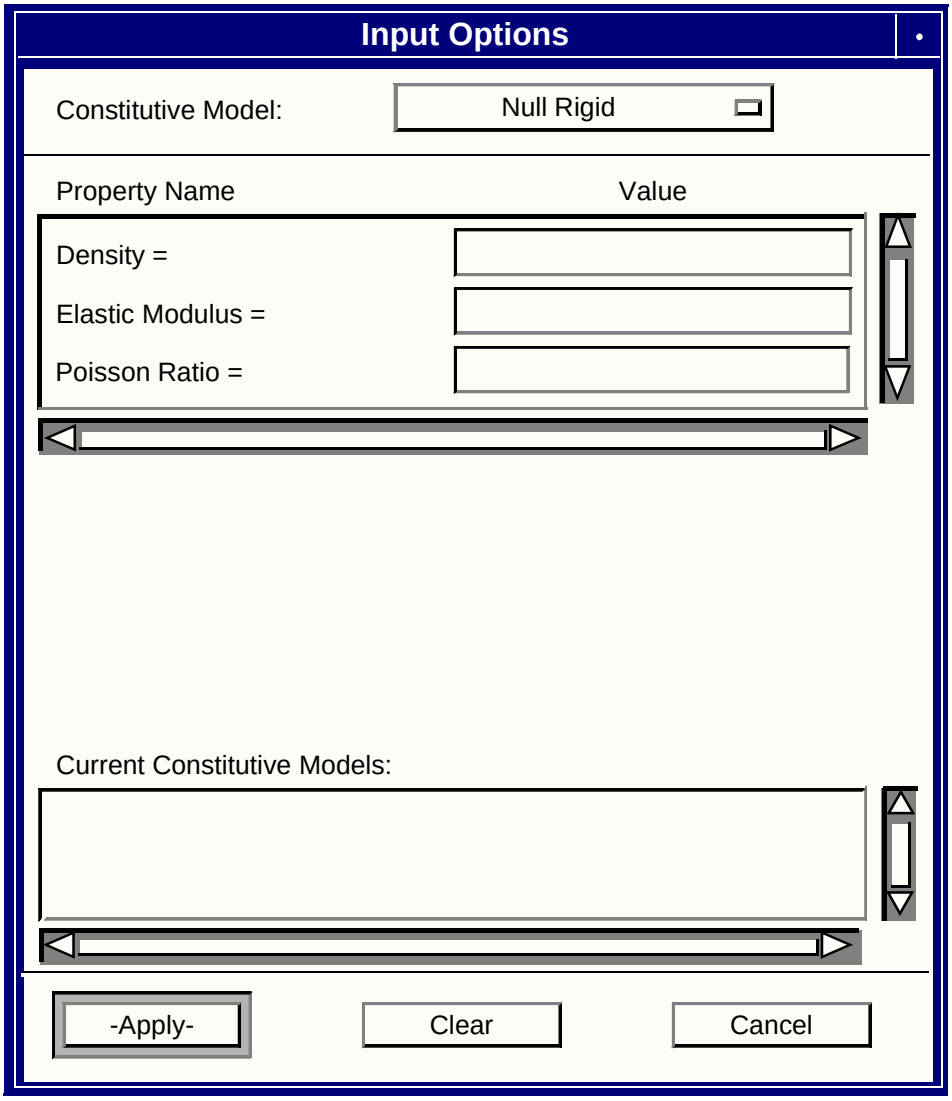
Viscoelastic

This subordinate form appears when the Input Properties button is selected on the Materials form when the Viscoelastic Constitutive model is selected. Use this form to define the data for PAMCRASH Material Type 5.



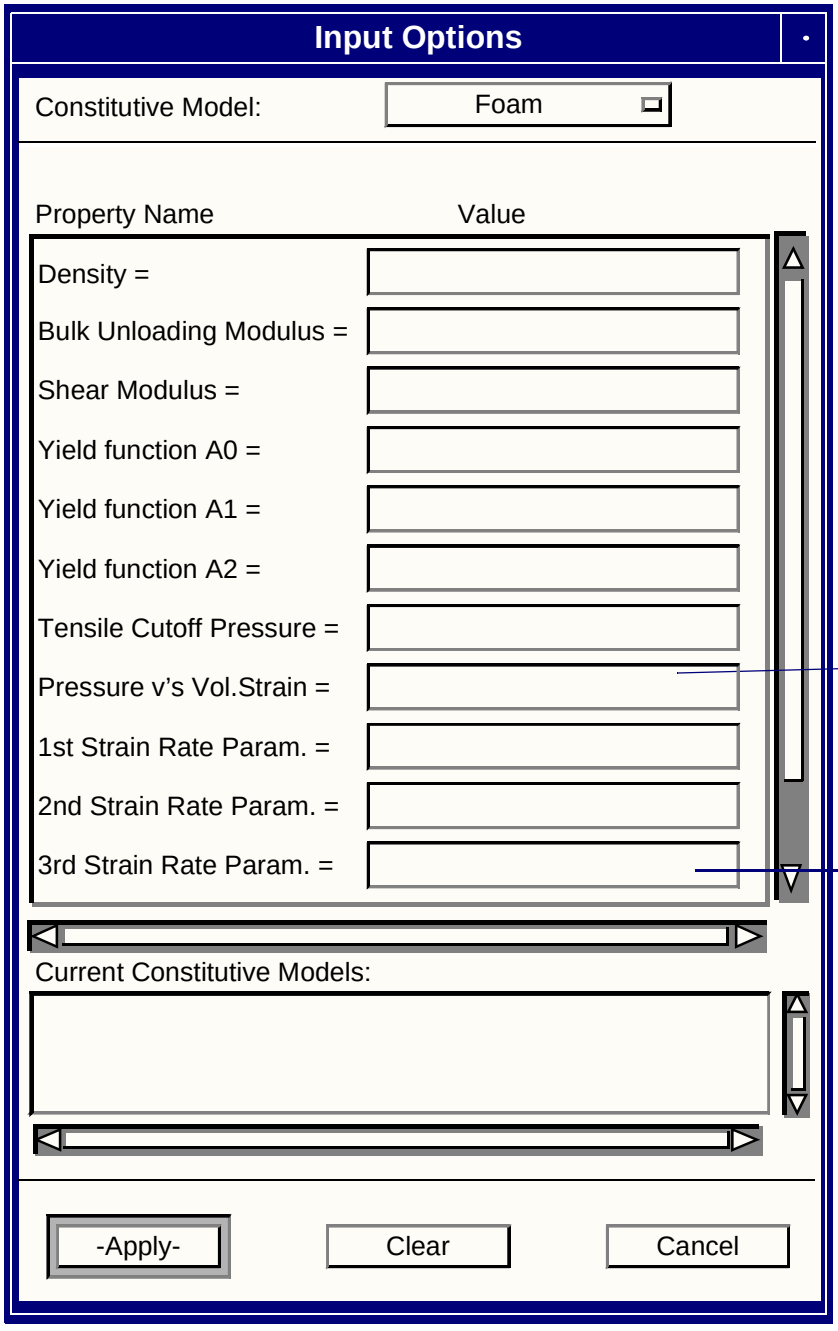
Null Rigid

This subordinate form appears when the Input Properties button is selected on the Materials form when the Null Rigid Constitutive model is selected. Use this form to define the data for PAMCRASH Material Types 99, 100 and 200.



Foam

This subordinate form appears when the Input Properties button is selected on the Materials form when the Foam constitutive model is selected.Use this form to define the data for PAMCRASH Material Type 2.



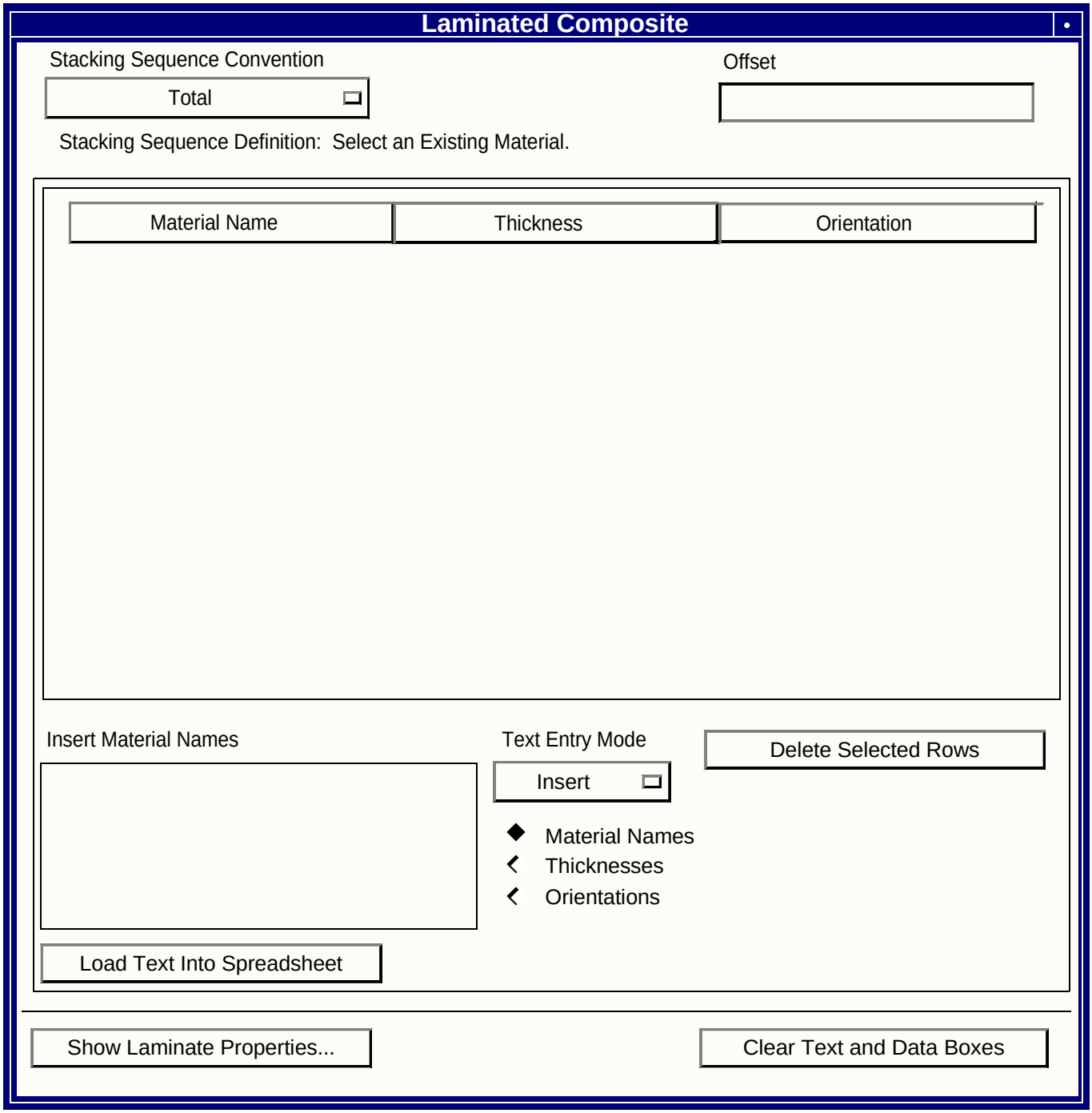
Note that this requires a strain dependent field. This field can have up to 10 pairs of datapoints.

Additional data list:
4th Strain Rate Parameter
5th Strain Rate Parameter
6th Strain Rate Parameter

Composite

Laminate

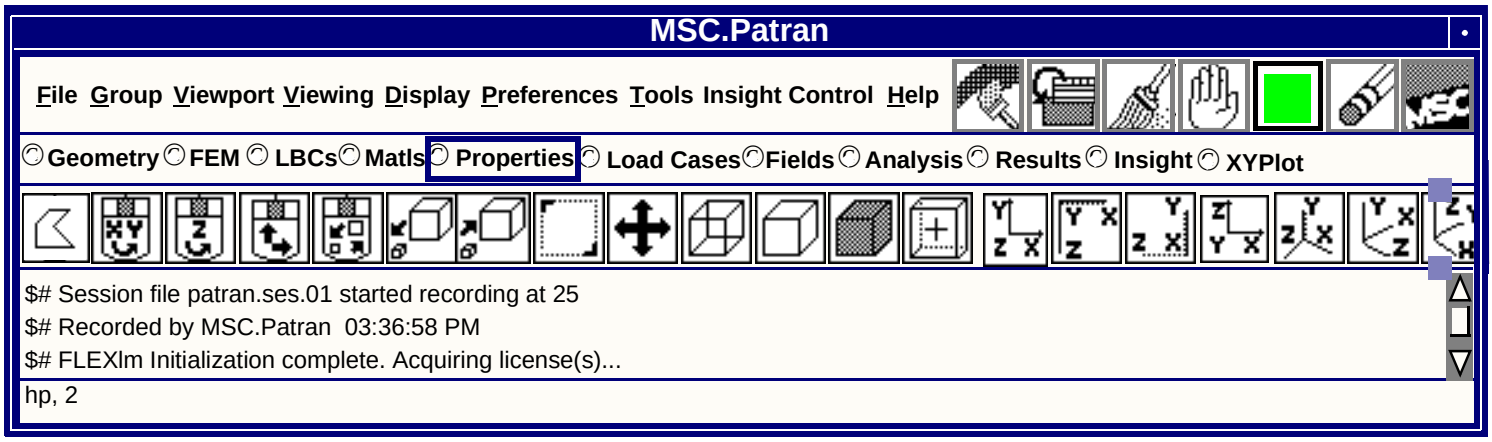
This subordinate form appears when Composite is the object on the Material form, and laminate is the selected method. Only PLY Model 0 (Unidirectional composite bi-phase ply model) is supported and only the element local coordinate system is supported for specifying ply orientation.



2.5 Element Properties

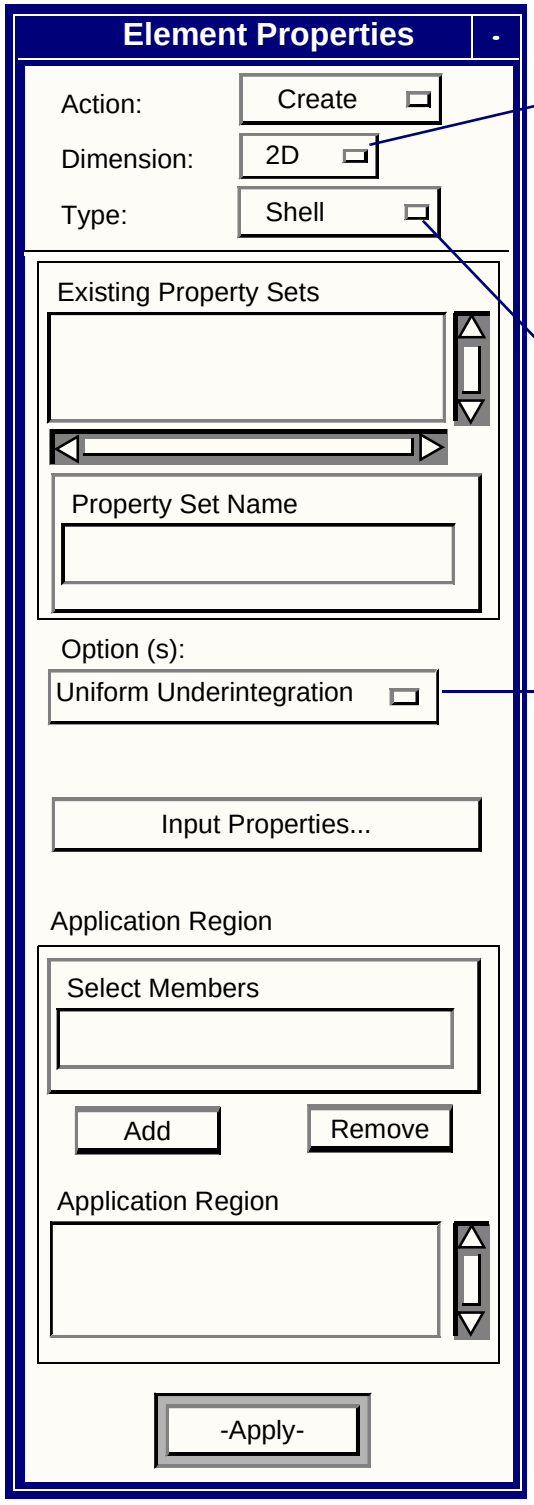
The Element Properties form appears when the Element Properties toggle, located on the MSC.Patran main form, is chosen.There are several option menus available when creating element properties. The selections made on the Element Properties form will determine which element property form appears, and ultimately, which PAMCRASH element will be created.

The following pages give an introduction to the Element Properties form, and details of all the element property definitions supported by the MSC.Patran PAMCRASH Preference.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Element Properties Form

This form appears when Element Properties is selected on the main form. There are *four option menus* on this form (under Dimension), each will determine which PAMCRASH element type will be created, and which property forms will appear. The individual property forms are documented later in this section. For a full description of this form, see **Element Properties Forms** (p. 41) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments.*



Use this option menu to define the elements dimension. The options are:

0D (point elements)

1D (bar elements)

2D (tri and quad elements)

3D (tet, wedge, and hex elements)

This option menu depends on the selection made in the Dimension option menu. Use this menu to define the *general type of element*.

This option menu *may* or *may not* be presented, and its contents depend heavily on the selections made in Dimension and Type. See for more help.

The following table outlines the *option menus* when Analysis Type is set to *Structural*. Note that not all material types are supported for all properties. This is a function of PAMCRASH.

Table 2-5 Structural Options

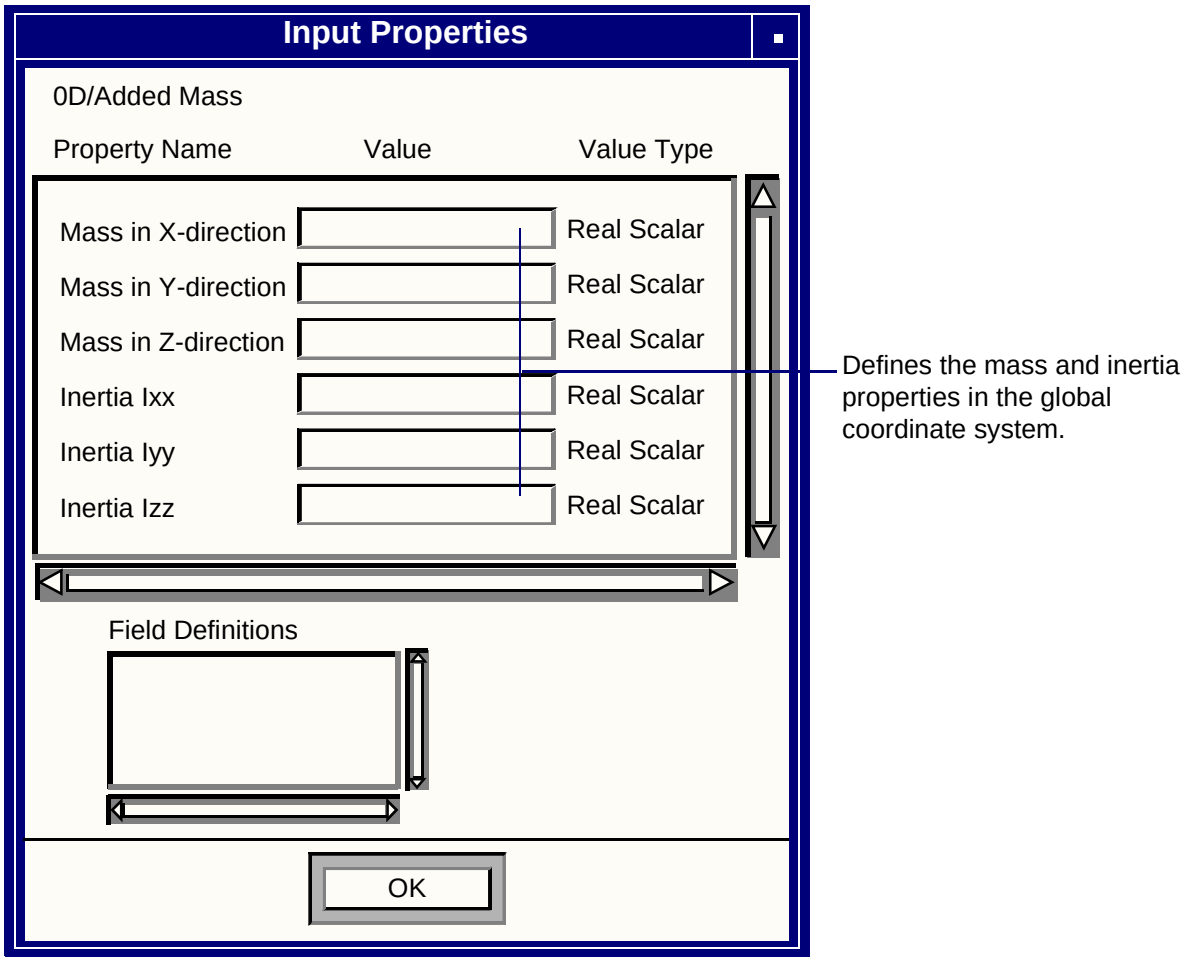
Degree	Type	Option 1	Option 2
0D	❑ 0D Mass		
1D	❑ Beam		
	❑ Rod		
	❑ Spring		
2D	❑ Shell	❑ Homogeneous	Uniform Underintegration
			Hughes-Tezduyar
		❑ Laminate	Uniform Underintegration
			Hughes-Tezduyar
3D	❑ Solid	❑ H'glass Viscous Base Uniform Underintegration	
		❑ H'glass Viscous Shape Selective Reduced Integration	
		❑ H'glass Stiffness Shape	

0D Mass

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option(s)	Topologies
Create	0D	Mass		Point

Use this form to create a **MASS__/_** *data entry*. This defines added mass for the structural model.

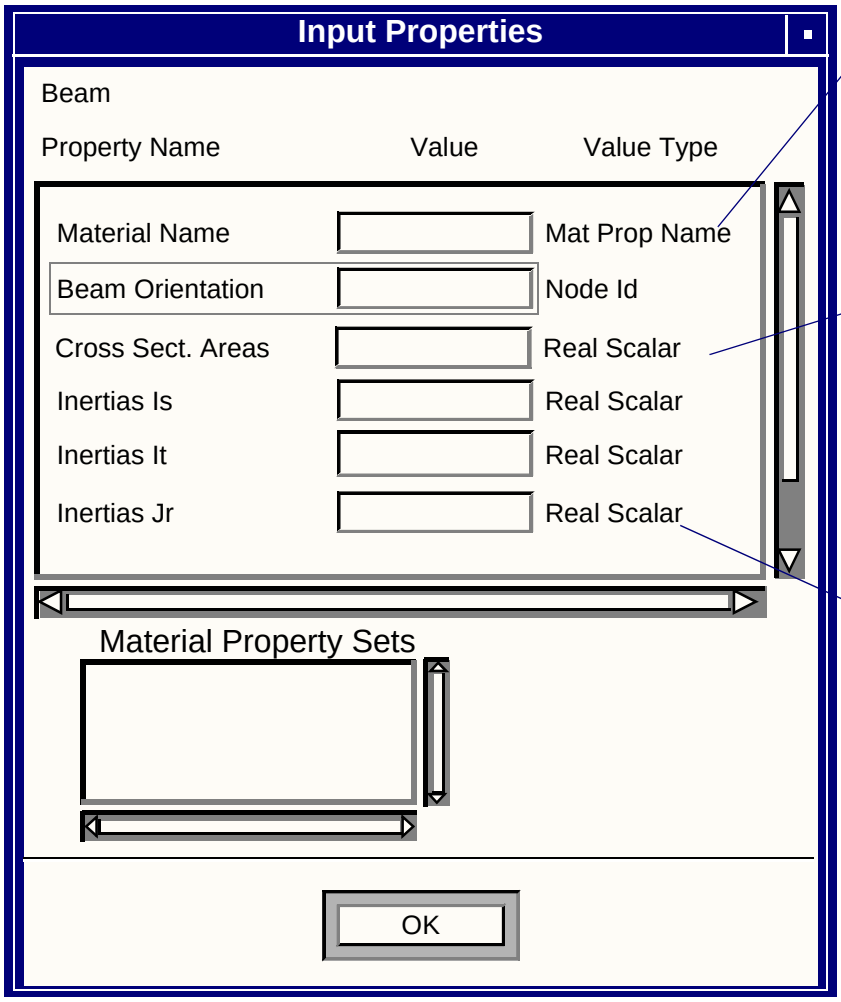


Beam

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option(s)	Topologies
Create	1D	Beam		Bar / 2

Use this form to create a **BEAM___/_** *entry*. The area and other data is written to a **Material Type 200/201/212** record.



Defines the material to be used. A list of all materials, currently in the database, is displayed when data is entered. Either select from the list using the mouse, or type in the name. This property is required.

Defines the cross-sectional area of the element. These values can either be real values, or references to existing field definitions. This property is required. Use the field definition with care as this can result in a material record for each element.

Additional data here are:

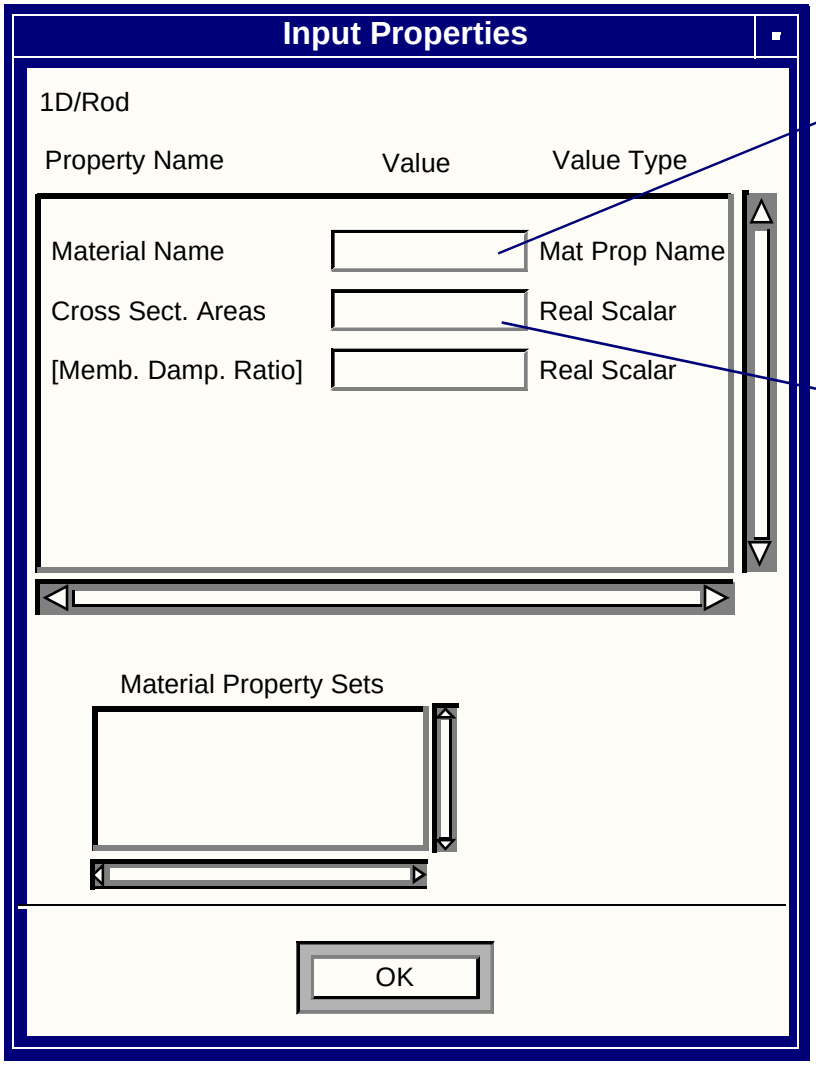
1. Shear Effective Area
2. [Memb. Damp. Ratio]
3. [Bend.Damp. Ratio]
4. [Torsion Damp. Ratio]

Rod

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option(s)	Topologies
Create	1D	Rod		Bar / 2

Use this form to create a **BAR___/_** *entry*. The area and membrane damping data is written to a **Material Type 200/201/202** record.



Defines the material to be used. A list of all materials, currently in the database, is displayed when data is entered. Either select from the list using the mouse, or type in the name. This property is required.

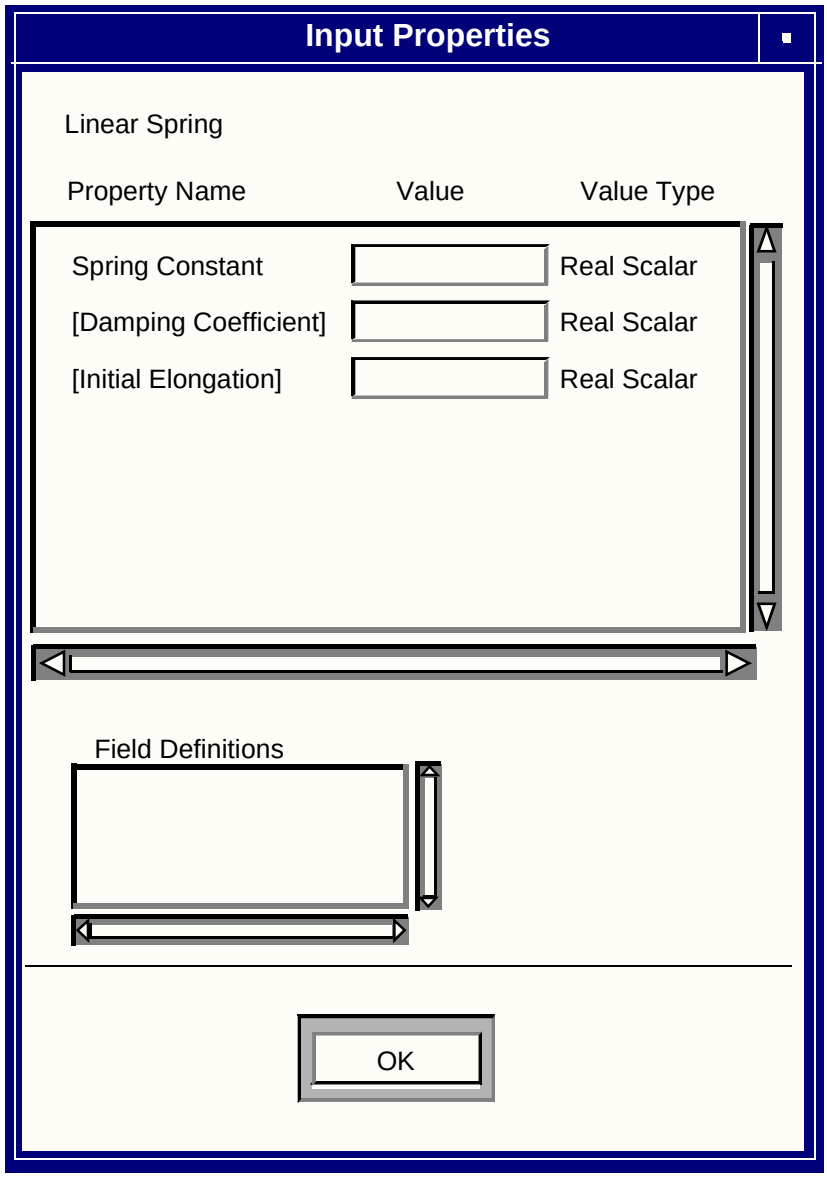
Defines the cross-sectional area of the element. These values can either be real values, or references to existing field definitions. This property is required. Use the field definition with care as this can result in a material record for each element.

Linear Spring

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option(s)	Topologies
Create	1D	Spring		Bar / 2

Use this form to create a **BAR___/_** *and a simplified* **Material Type 204** *entry*.

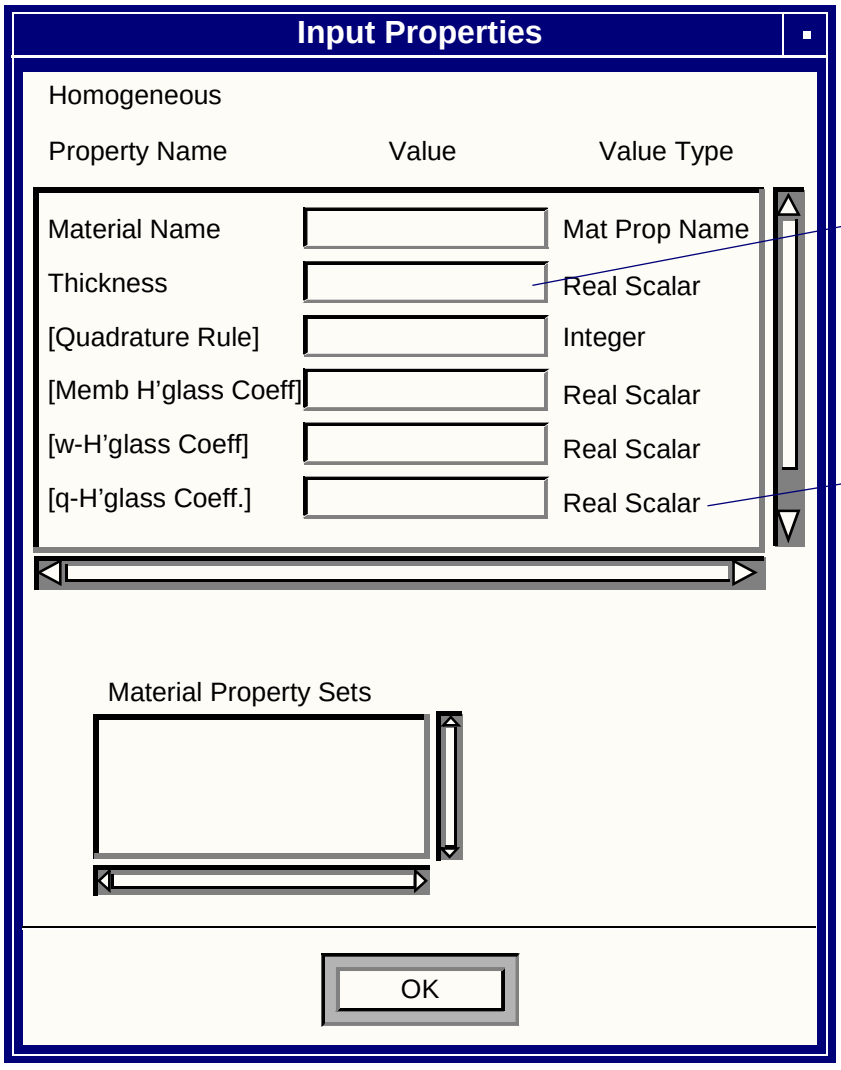


Shell

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option1	Option(2)	Topologies
Create	2D	Shell	Homogeneous	Uniform Underintegration Hughes-Tezduyar	Tria/3, Quad/4
			Laminate	Hughes-Tezduyar Uniform Underintegration	

Use this form to create a **SHELL___/_** entry. The data is written to a **Material Type 100/101/102/103/105/106** record.



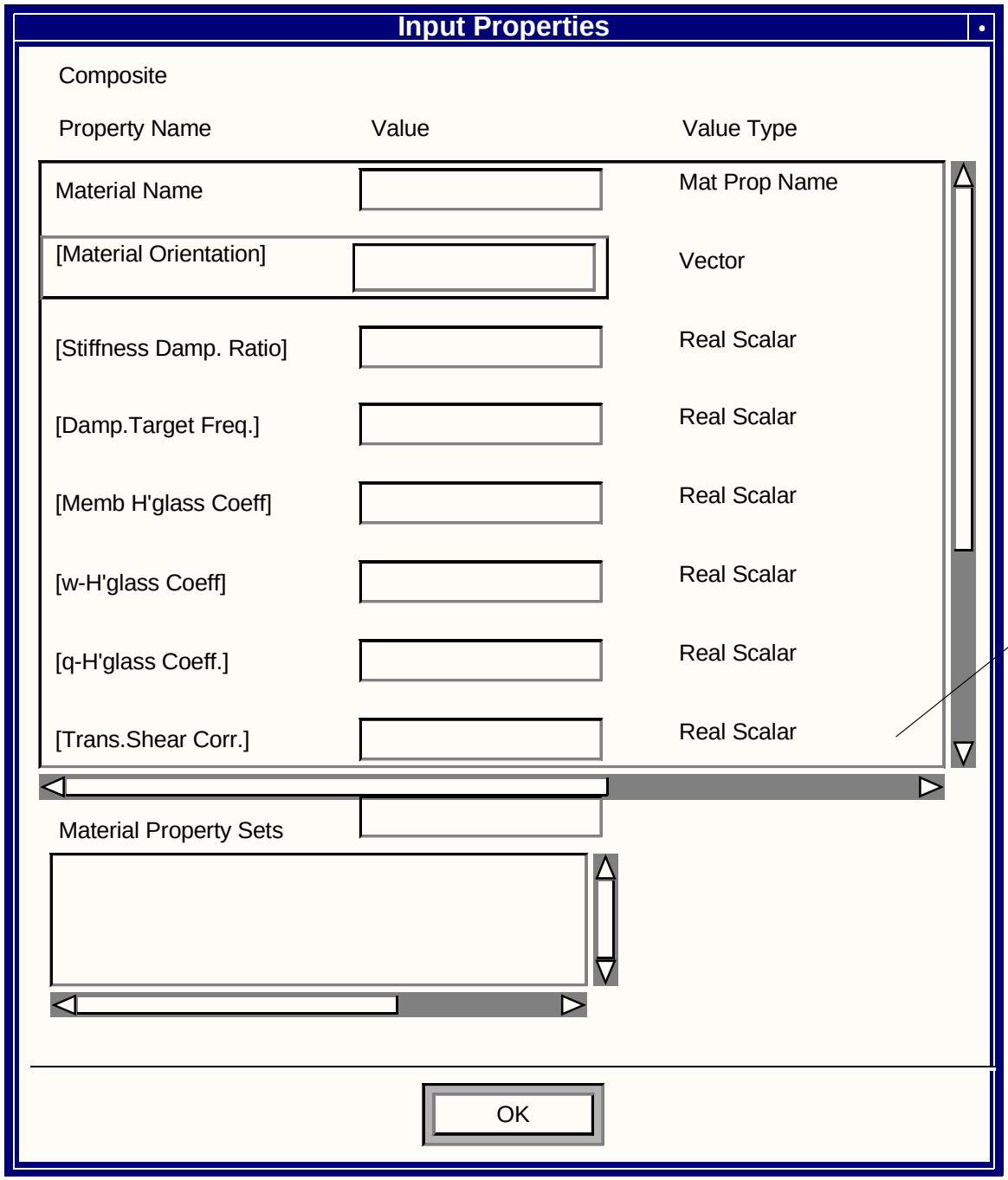
Defines the thickness which will be uniform over each element. This value can either be real or a reference to an existing field definition.

Additional data here are:

1. [Trans. Shear Corr.]
2. [Stiffness Damp. Ratio]
3. [Damp. Target Freq.]
4. [Deletion Plastic Strain]
5. [Minimum Deletion Time]

Note: No orientation section is required as only laminated shells have directional properties. Note also that both shell options (Uniform Underintegration / Hughes-Tezduyar) require the same data.

Use this form to create a SHELL_ _/_ entry. The data is written to a Material Type 130 record.

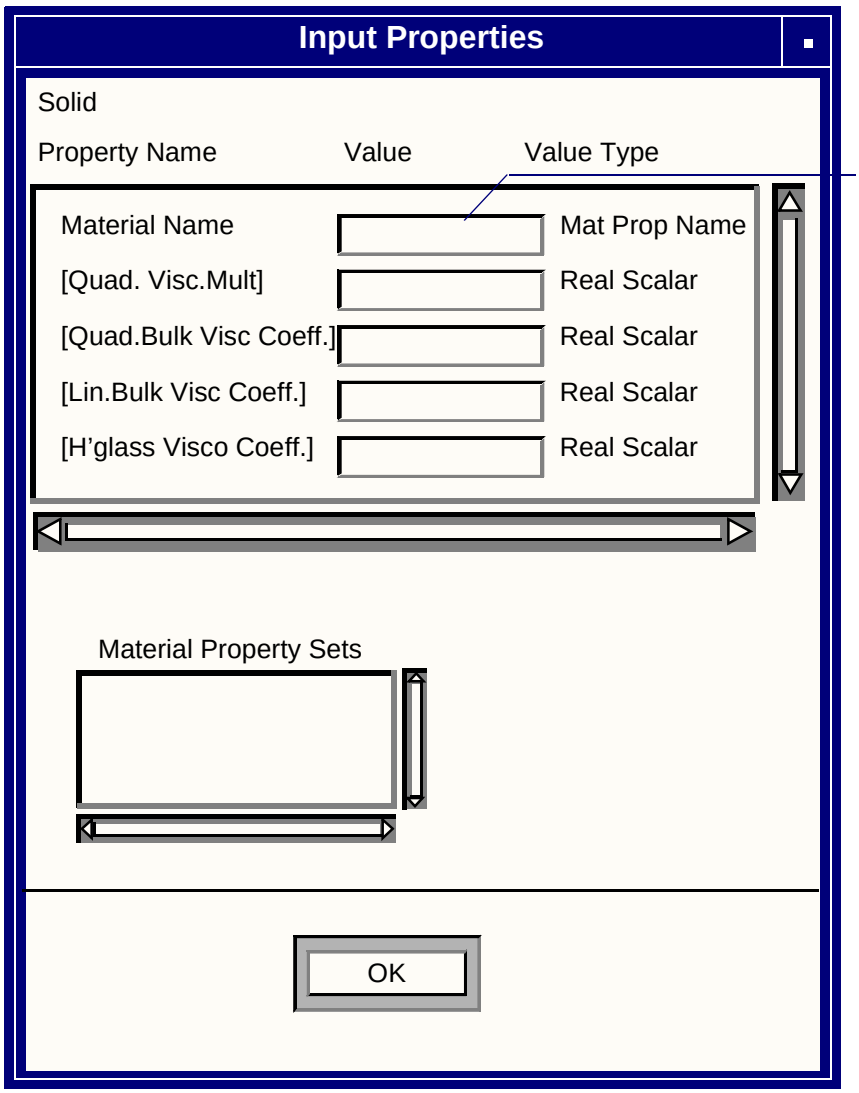


Solid

This subordinate form appears when the Input Properties button is selected on the Element Properties form when the following options are chosen.

Action	Dimension	Type	Option 1	Option 2	Topologies
Create	3D	Solid	Hourglass Viscous Base	Uniform Underintegration	Tet/4, Wedge/6 Hex/8
				Selective Reduced Integration	
			Hourglass Viscous Shape	Uniform Underintegration	
				Selective Reduced Integration	
			Hourglass Stiffness Shape	Uniform Underintegration	
				Selective Reduced Integration	

Use this form to create a **SOLID_/_** entry. The data is written to a **Material Type 1/2/5/99** record.

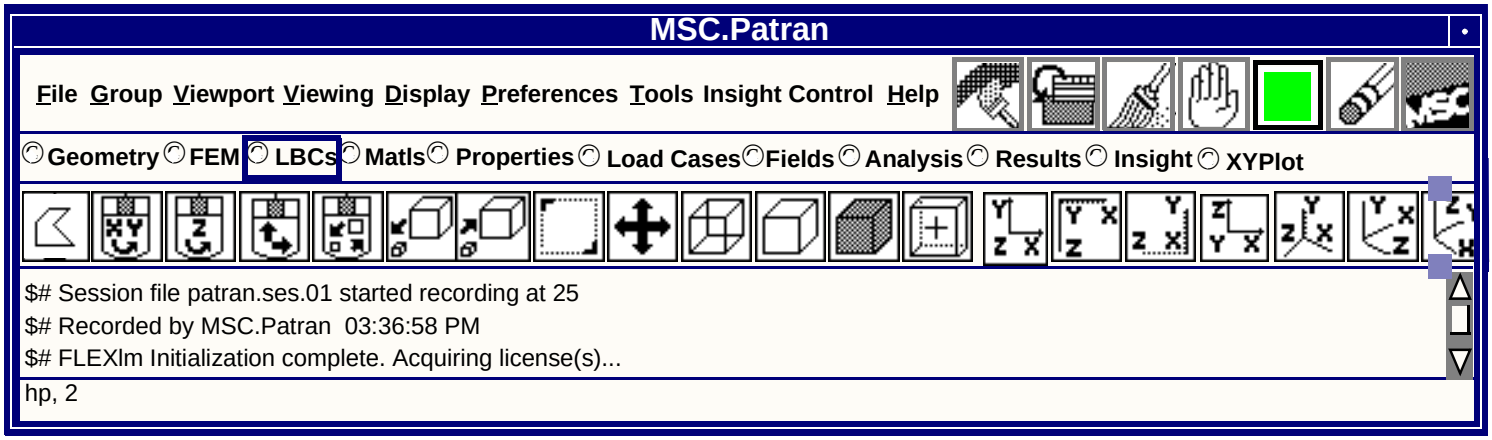


Defines the material to be used. A list of all materials, currently in the database, is displayed when data is entered. Either select from the list using the mouse, or type in the name.

2.6 Loads and Boundary Conditions

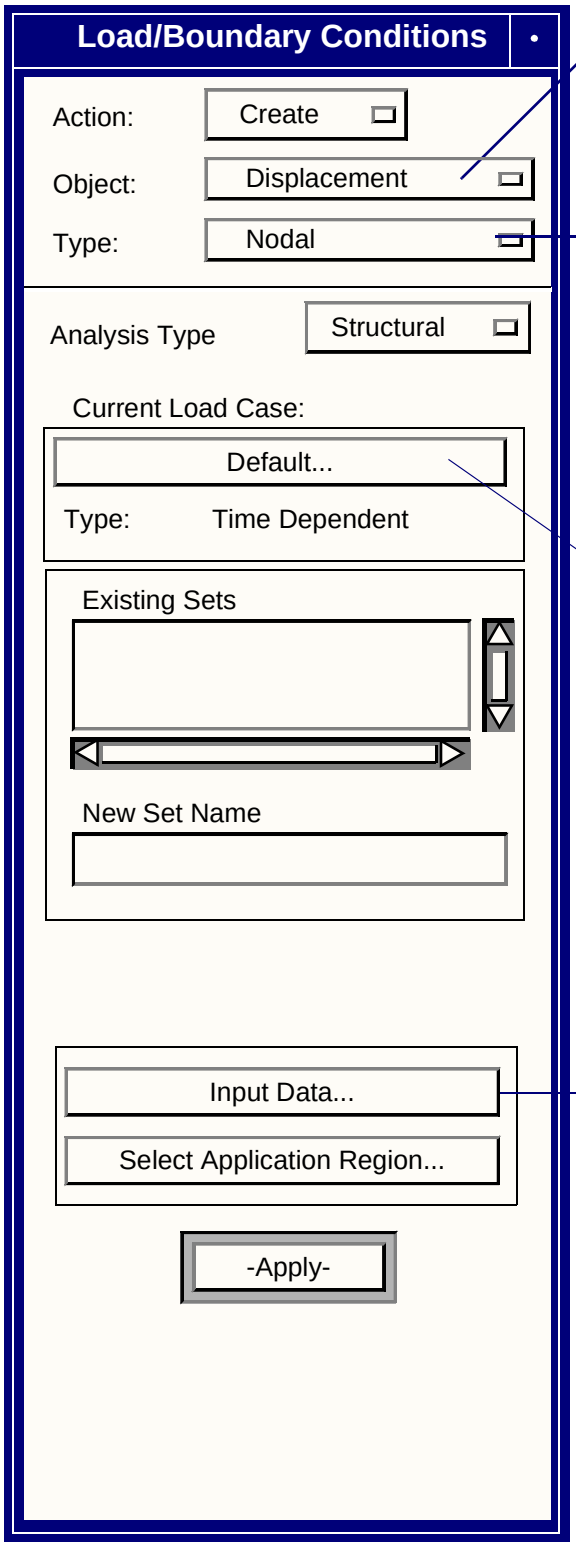
The Loads and Boundary Conditions form will appear when the Loads/BCs toggle, located on the MSC.Patran application selections, is chosen. When creating loads and boundary conditions there are several option menus. The selections made on the Loads and Boundary Conditions menu will determine which loads and boundary conditions form appears, and ultimately, which PAMCRASH loads and boundary conditions will be created.

The following pages give an introduction to the Loads and Boundary Conditions form, and details of all the loads and boundary conditions supported by the MSC.Patran PAMCRASH Analysis Preference.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Loads & Boundary Conditions Form

This form appears when Loads/BCs is selected on the main form. The Loads and Boundary Conditions form is used to provide options to create the various PAMCRASH loads and boundary conditions. For a definition of full functionality, see **Loads and Boundary Conditions Form** (p. 18) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments.*



Defines the general load type to be applied. Object choices are Displacement, Force, Initial Velocity, Velocity, Contact, Geometric Rigid Wall, Planar Rigid Wall, Nodal Rigid Body, Nodal Constraint, Nodal Damping.

Defines what type of region is to be loaded. The available options here depends on the selected Object. The general selections can be Nodal or Element Uniform. *Nodal* is applied explicitly to nodes. *Element Uniform* defines a constant value to be applied over an entire element, element face, or element edge.

Current Load Case type is set on the Load Case menu. When the Load Cases toggle, located on the main form, is chosen the Load Cases menu will appear. Under Load Case Type, select either *Static* or *Time Dependent*, then enter the name of the case, and click on the apply button.

Generates Transient Input Data form.

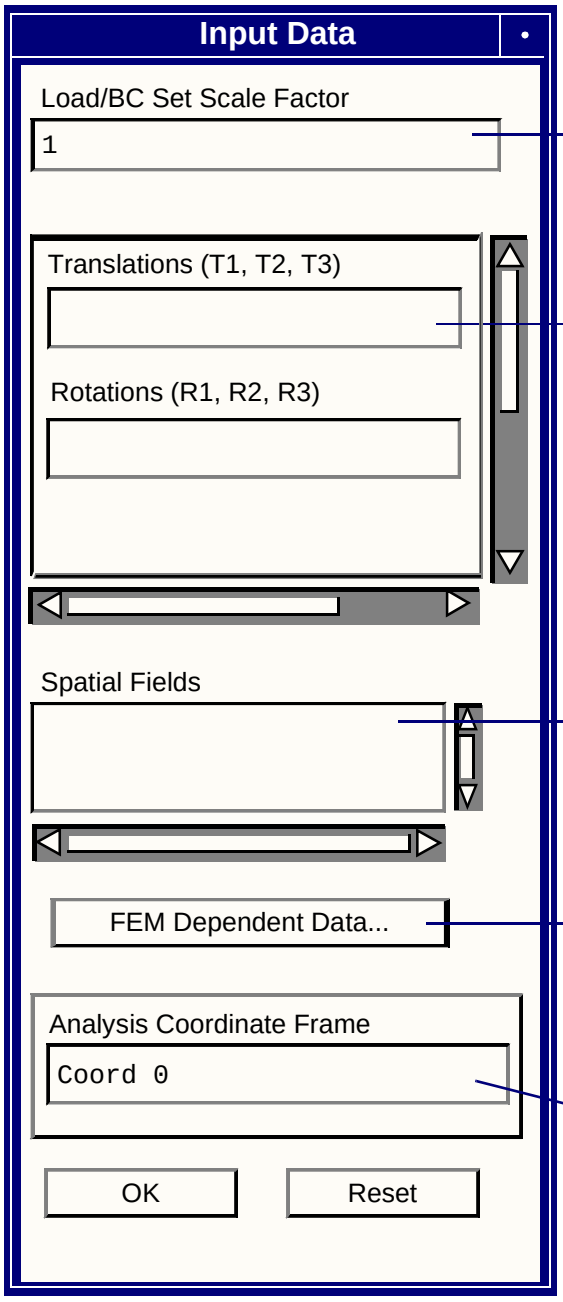
The following table outlines the options when Create is the selected action.

Table 2-6 Loads and Boundary Condition Objects

Object	Type
❑ Displacement	Nodal
❑ Force	Nodal
❑ Initial Velocity	Nodal
❑ Velocity	Nodal
❑ Contact	Element Uniform
❑ Geometric Rigid Wall	Nodal
❑ Planar Rigid Wall	Nodal
❑ Nodal Rigid Body	Nodal
❑ Nodal Constraint	Nodal
❑ Nodal Damping	Nodal

Static (Not Time Varying)

This subordinate form appears when the Input Data button is selected on the Loads and Boundary Conditions form when the Current Load Case Type is *Static*. The Current Load Case Type is set on the Load Case form, for more information see **Loads and Boundary Conditions Form** (p. 18) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments*. The information on the Input Data form will vary depending on the selected Object. Defined below is the standard information found on this form. Note that this form is not used with the Pamcrash Preference.



Defines a general scaling factor for *all values* defined on this form. The default value is 1.0. Primarily used when field definitions are used to define the load values.

Input Data in this section *will vary*. See **Object Tables** (p. 47) for detailed information.

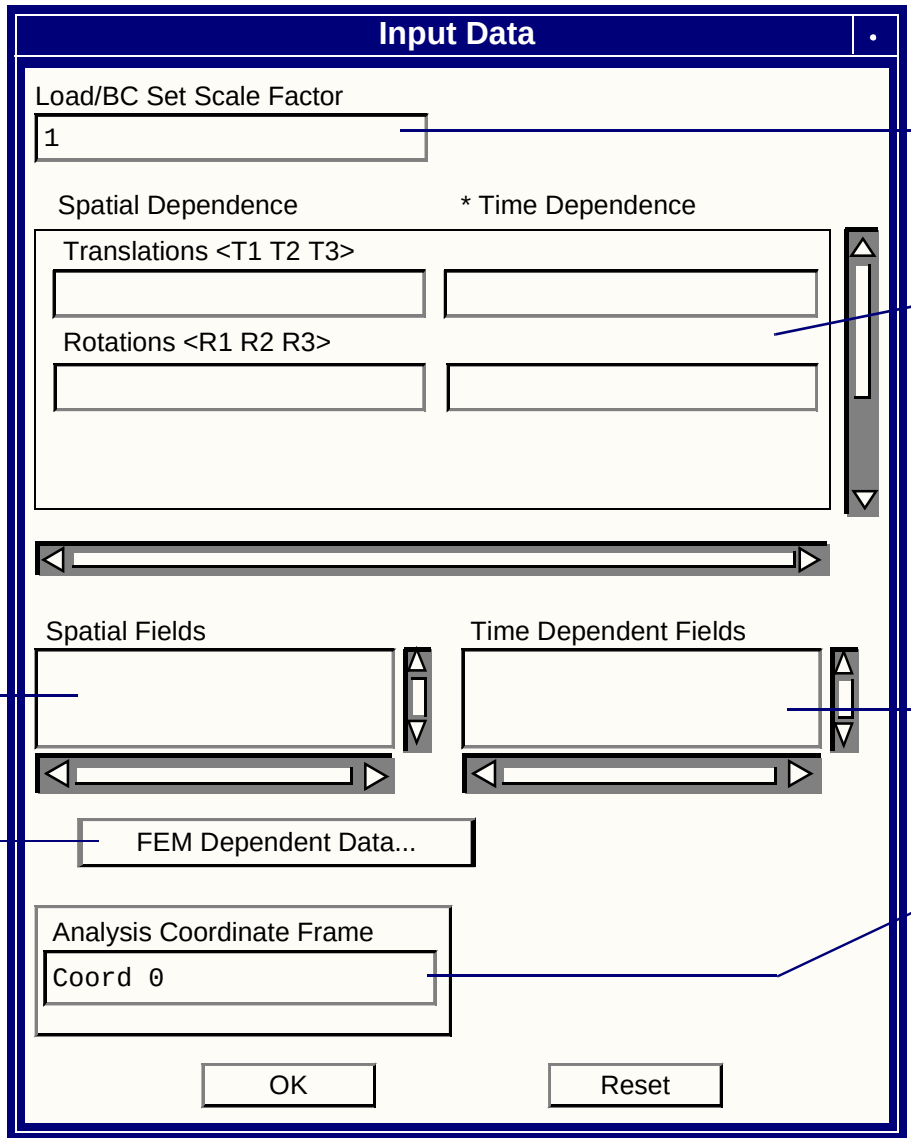
When specifying real values in the Input Data entries, spatial fields *can be* referenced. All defined spatial fields *currently* in the database are listed. If the input focus is placed in the Input Data entry, and a spatial field is selected by double clicking in this list, a reference to that field will be entered in the Input Data entry.

Displays a Discrete FEM Fields input form to allow field creation and modification within the loads/bcs application. Visible only when focus is set in a databox which can have a DFEM field reference.

Defines the coordinate frame used to interpret the degree-of-freedom data defined on this form. *This only appears on the form for Nodal type loads*. This can be a reference to any existing coordinate frame definition.

Time Dependent (Time Varying)

This subordinate form appears when the Input Data button is selected on the Loads and Boundary Condition form when the Current Load Case Type is *Time Dependent*. The Current Load Case Type is set on the Load Case form, for more information see **Loads and Boundary Conditions Form** (p. 18) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments* and **Load Cases** (p. 55). The information on the Input Data form will vary, depending on the selected Object. However, it should be noted that not all LBC Objects support time dependence. Defined below is the standard information found on this form.



Defines a general scaling factor for *all values* defined on this form.The default value is 1.0. Primarily used when field definitions are used to define the load values.

Input Data in this section *will vary*. See **Object Tables** (p. 47) for detailed information.

When specifying time dependent values in the Input Data entries, time dependent fields can be referenced. *All* defined time dependent fields *currently* in the database are listed. If the input focus is placed in the Input Data entry, and a time dependent field is selected by double clicking in this list, a reference to that field will be entered in the Input Data entry.

Defines the coordinate frame to be used to interpret the degree-of-freedom data defined on this form. This *only appears on the form for Nodal type loads*. This can be a reference to any existing coordinate frame definition.

Displays a Discrete FEM Fields input form to allow field creation and modification within the loads/bcs application. Visible only when focus is set in a databox which can have a DFEM field reference.

When specifying real values in the Input Data entries, spatial fields can be referenced. *All* defined spatial fields *currently* in the database are listed. If the input focus is placed in the Input Data entry, and a spatial field is selected by double clicking in this list, a reference to that field will be entered in the Input Data entry.

Contact Toolkit

This section describes the user interface provided by MSC.Patran to access the contact features of explicit dynamics finite element codes. This interface is used during definition of the contact LBC types: Self Contact, Subsurface, Master-Slave Surface, and Master-Slave Node.

Tools have been provided to enable the user to quickly and easily define contact conditions. Specification of contact is conceptually simple, involving either one or two contact surfaces, and a set of contact parameters which control the interaction of the surfaces.

Contact Types. A contact condition in which a single logical surface may come into contact only with itself is described as self-contact, and requires the specification of a single Application Region. A contact condition in which two logical surfaces may contact each other is described as Master-Slave contact, and requires specification of two Application Regions. Master-Slave contact is further subdivided by the definition of Master-Slave Surface and Master-Slave Node. Master-Slave Surface describes the condition in which both the master and slave surfaces are defined using element faces, whereas Master-Slave Node describes the condition where the Slave surface is described only using nodes.

Contact Construction. Tools are provided to enable the construction of contact surfaces, using the standard MSC.Patran select tool mechanisms (2D elements, 3D element faces), or groups. Contact subsurfaces can also be constructed using these tools, and later used to define a complete logical contact surface. This functionality allows the user to use the select tool to specify application regions on MSC.Patran geometry or the associated FEM entities or to define a more complex contact surface that is assembled from a mixture of 2D and 3D element faces, and to simply combine groups of 2D elements taking into account the direction of the contact outward normal. (For 2D elements, the outward normal can be reversed for contact purposes without modifying the underlying element topology.) Use of the group select mechanism is restricted to FEM entities only. Visualization of the specified contact condition is provided by graphically previewing but is not currently supported for geometry entities.

“Simple” contact surfaces include surfaces which may be described entirely by the faces of 3D elements, or by 2D elements whose outward normals are aligned with the desired contact normal direction. These contact surfaces may be constructed entirely using a single select mechanism (either Select Tool or Group method). Simple contact surfaces may not include a mixture of 3D element faces and 2D elements, or 2D elements whose outward normals are not all aligned with the desired contact normal direction.

“Complex” contact surfaces are defined as those surfaces which consist of a mixture of 2D elements and 3D element faces, or all 2D elements but with some of the outward normals incorrectly aligned. Contact conditions which include complex contact surfaces must be constructed using “Subsurfaces,” where each subsurface is a “Simple” contact surface. Definition of contact surfaces is limited to one method, i.e. it is not permissible to mix “Select Tool,” “Group,” or “Subsurface” within the definition of a contact surface.

Use of the Select Tool. The select tool is used to graphically select the desired entities from the model. When this method is selected, the user must specify which dimensionality the intended object has, i.e. 3D, 2D or Nodal. If the selected dimensionality is 2D, then the user can further specify whether the top, bottom or both surfaces option is required. Selection of top will result in a contact surface whose outward normal is coincident with the element outward normal, whereas selection of bottom will result in a contact surface whose outward normal is in the opposite direction to the element outward normal. The user can toggle between Top, Bottom or Both at any time during selection; however, all of the selected entities will be assigned the same logical direction. Selection of 3D allows the user to select either all or all free faces of 3D elements.

No user specification of the contact normal direction is required for 3D elements since the program automatically specifies this direction. No contact direction is applicable to Nodal contact surfaces.

It is not permissible to mix 3D, 2D and Nodal entities within a single Application Region. (This functionality is provided through the use of contact subsurfaces.) The select tool can be used to select on the basis of either FEM or Geometry entities.

Use of the Group Tool. The Group tool is used to define simple contact surfaces on the basis of MSC.Patran group names. When this method is selected, the user must specify which dimensionality the intended object has, i.e. either 3D, 2D or Nodal. The entities which will be selected for use in the contact surface in this case are either all 3D free surfaces in the group, all 2D elements or all nodes contained in the selected group. In the case of 2D elements, the user may specify whether the contact normal direction is coincident with the element top, bottom or both faces. Multiple groups may be selected. However, it should be noted that both the selected element dimensionality and contact normal direction apply across all selected groups.

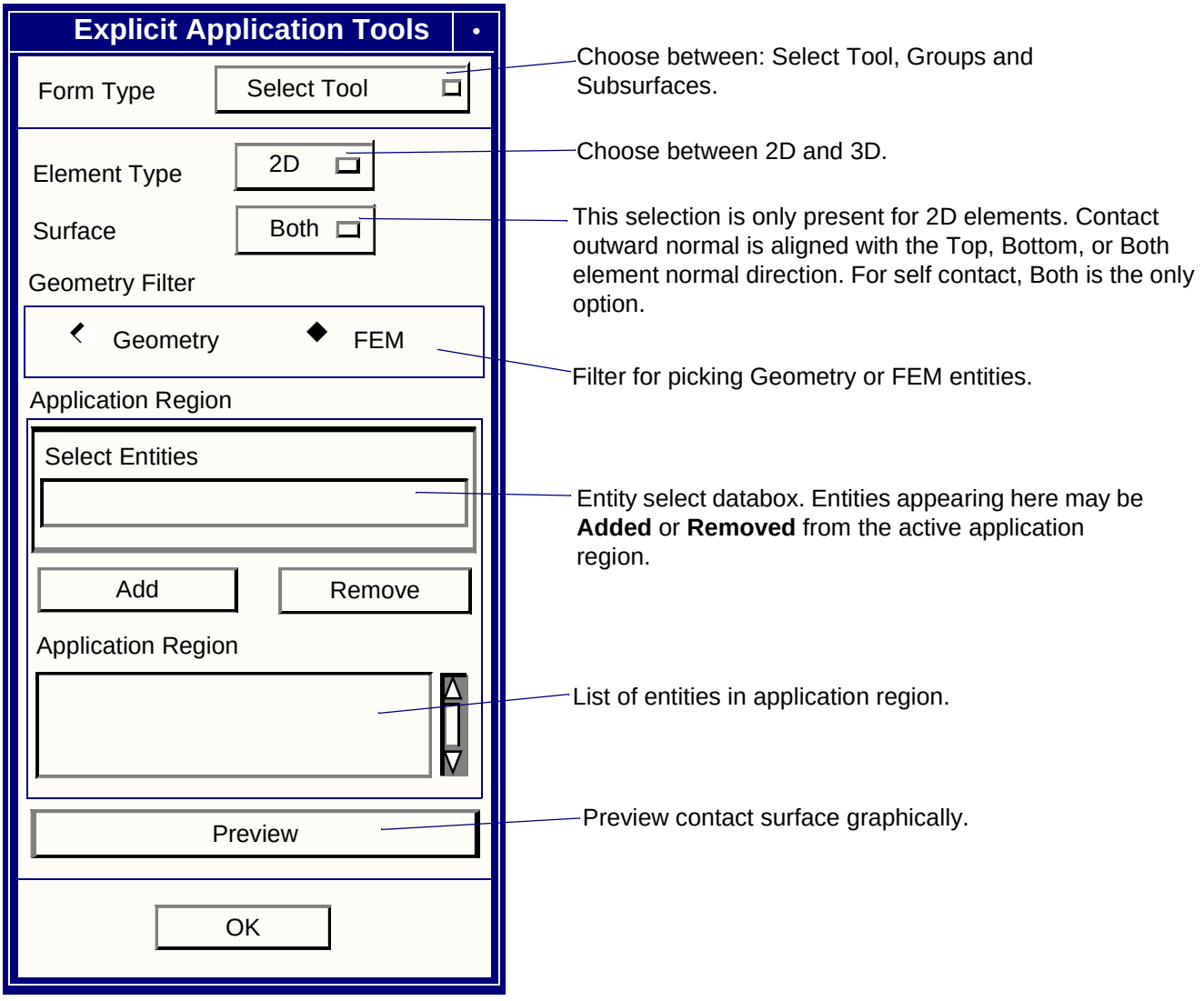
Use of the Subsurface Tool

Contact Subsurfaces may be defined using either of the above methods. Subsurfaces may then be used in the specification of Master, Slave or Self contact surfaces. When this option is used, the user may not specify element dimensionality or contact normal direction since this information has already been defined during subsurface definition. As many sub-surfaces as required may be selected to form the desired complex contact subsurface.

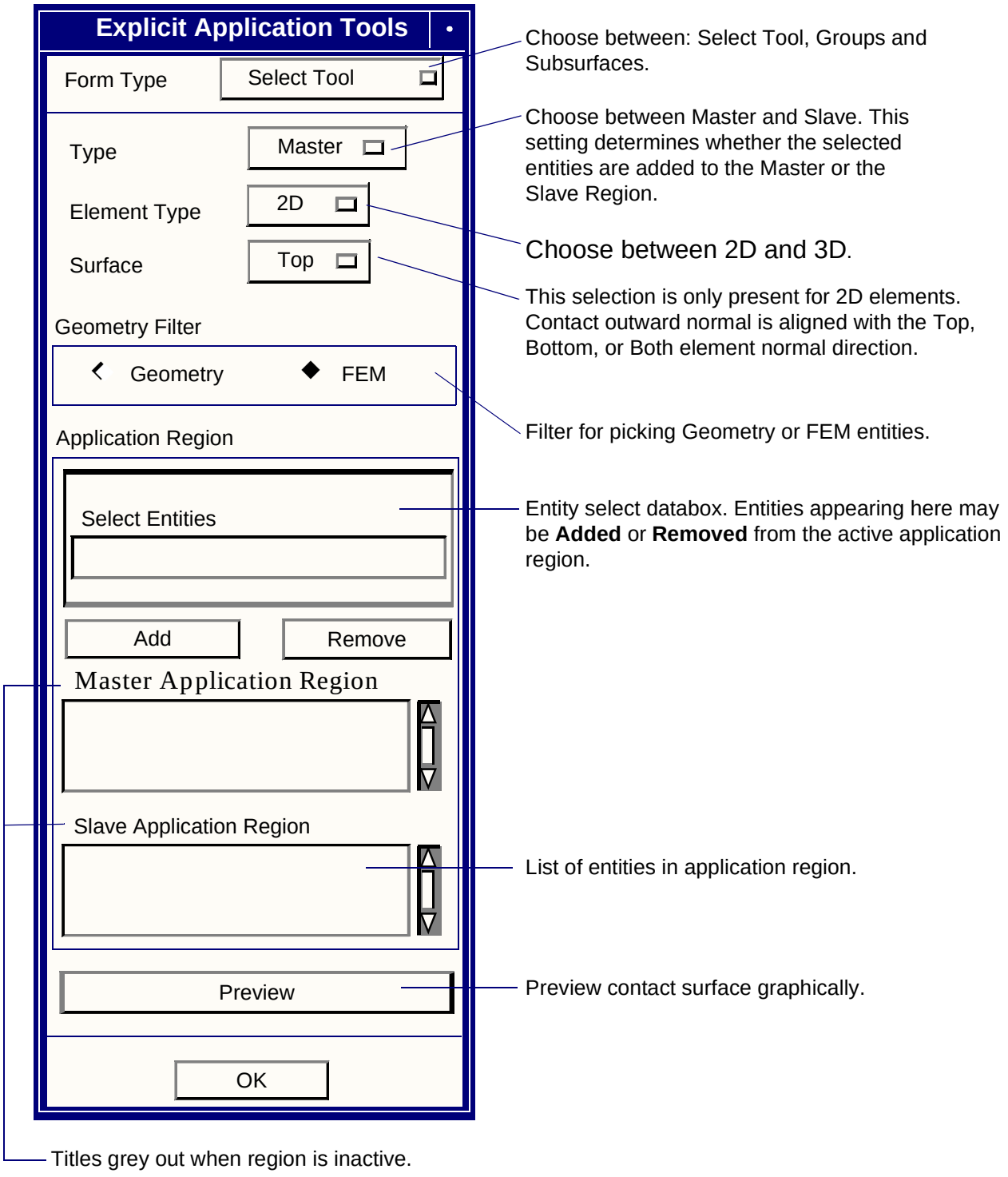
Application Region

This form is used to define contact surfaces. The form will vary depending upon which options are selected; however, two basic configurations are used depending on whether the contact condition requires specification of a single contact surface or two contact surfaces.

Single Application Region. The following form is used to define a single surface contact or a subsurface.

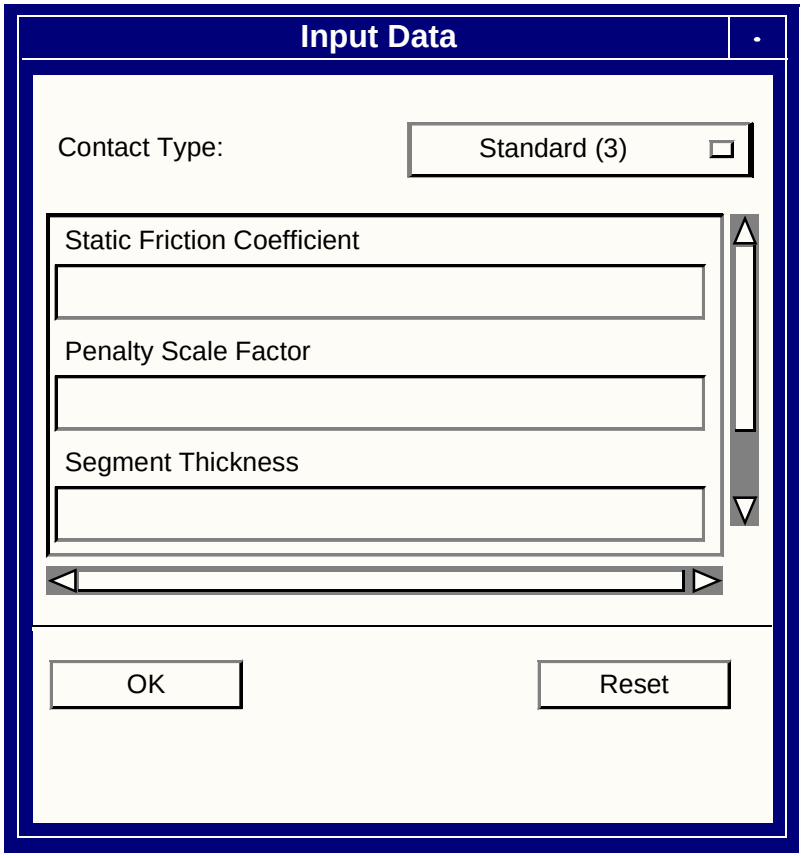

Explicit Application Tools
Form Type
Select Tool
Element Type
2D
Surface
Both
Geometry Filter
Geometry
FEM
Application Region
Select Entities
Add
Remove
Application Region
Preview
OK
Choose between: Select Tool, Groups and Subsurfaces.
Choose between 2D and 3D.
This selection is only present for 2D elements. Contact outward normal is aligned with the Top, Bottom, or Both element normal direction. For self contact, Both is the only option.
Filter for picking Geometry or FEM entities.
Entity select databox. Entities appearing here may be Added or Removed from the active application region.
List of entities in application region.
Preview contact surface graphically.

Dual Application Region. The following form is used to define either of the master slave contact types.


Explicit Application Tools
Form Type
Select Tool
Type
Master
Element Type
2D
Surface
Top
Geometry Filter
Geometry
FEM
Application Region
Select Entities
Add
Remove
Master Application Region
Slave Application Region
Preview
OK
Choose between: Select Tool, Groups and Subsurfaces.
Choose between Master and Slave. This setting determines whether the selected entities are added to the Master or the Slave Region.
Choose between 2D and 3D.
This selection is only present for 2D elements. Contact outward normal is aligned with the Top, Bottom, or Both element normal direction.
Filter for picking Geometry or FEM entities.
Entity select databox. Entities appearing here may be **Added** or **Removed** from the active application region.
List of entities in application region.
Preview contact surface graphically.
Titles grey out when region is inactive.

Input Data

The Input Data form is used to specify parameters which control the behavior of the contact condition. The contents of the form will vary depending upon which option is selected. No Input Data is required for the Subsurface option since subsurfaces do not constitute a contact condition on their own.



Object Tables

There are areas on the static and transient input data forms where the load data values are defined. The data fields which appear depend on the selected load Object and Type. In some cases, the data fields also depend on the selected Target Element Type. The following Object Tables outline and define the various input data that pertains to a specific selected object:

Displacement

Object	Type	Analysis Type
Displacement	Nodal	Structural

This LBC type is used to define a **BOUNC_/_** entry. The optional rigid body information on this entry is not supported. The optional local coordinate definition is generated if a local coordinate system is selected (**FRAME_/_** entry). Time history information is ignored. The scale factor has no effect.

Input Data	Description
Translations (T1,T2,T3)	Enter 0 for a translational constraint and "," for translational freedom.
Rotations (R1,R2,R3)	Enter 0 for a rotational constraint and "," for rotational freedom.

Force

Object	Type	Analysis Type
Force	Nodal	Structural

This LBC type is used to define a **CONLO_/_** entry for concentrated loads on nodes. An auxiliary **FUNCT_/_**entry is defined from the time dependent field selected. The scale factor is used to scale the function, with default 1.0.Note that moments are not applicable.

Input Data	Description
Force (F1,F2,F3)	Defines the applied forces in the translational degrees-of-freedom, in the specified coordinate system.
Moment (M1,M2,M3)	Defines the applied moments in the rotational degrees-of-freedom.

Follower Force

Object	Type	Analysis Type	Dimension
Force	Element Uniform	Structural	2D / 3D

This LBC type is used to define a **CONLO_/_** entry for follower forces on a plane defined by three nodes. An auxiliary **FUNCT_/_** entry is defined from the time dependent field selected. The scale factor is used to scale the function, with default 1.0. Note that moments are not applicable. Note that Follower Forces are not supported for this MSC.Patran Pamcrash Preference version.

Input Data	Description
Force (F1)	Defines the applied force normal to the face of the 2D or 3D elements selected.

Initial Velocity

Object	Type	Analysis Type
Initial Velocity	Nodal	Structural

This LBC type is used to define a INVEL_/_ entry. The coordinate type will be cartesian unless a cylindrical axis is selected. Note that an initial velocity is required for every node in the model. Time history information is ignored.

Input Data	Description
Trans Veloc (v1,v2,v3)	Defines the V0 fields for translational degrees-of-freedom.
Rot Veloc (w1,w2,w3)	Defines the V0 fields for rotational degrees-of-freedom.

Velocity

Object	Type	Analysis Type
Velocity	Nodal	Structural

Is this LBC type is used to define one or more **VELBC_/_** entries. Displacement or Rotation Type 4 or 8 are used if a local coordinate is selected (but no FRAME_/_ entry is required). An auxiliary **FUNCT_/_** entry is defined from the time dependent field selected (these apply to all translational and all rotational degrees of freedom). The scale factor is used to scale the function, with default 1.0. Note that PAMCRASH only allows for a center of rotation at the global origin. However, local coordinate systems can be used to define the components of velocity.

Input Data	Description
Trans Veloc(v1,v2,v3)	Defines the enforced translational velocity values. These are in *model length units per unit time.*
Rot Veloc (w1,w2,w3)	Defines the enforced rotational velocity values. These are in *degrees per unit time.*

Contact

Object	Type	Analysis Type
Contact	Element Uniform	Structural

Four types of contact exist. Three of these are complete definitions and have associated input data. The fourth is the subsurface type which is used to define part of a contacting surface. This LBC type defines **SLINT_/_** and **SLINT2/_** entries. The following table outlines the options:

Table 2-7 Contact Type Options

Object	Option	Types
Contact	Self Contact Subsurface Master-Slave Surface Master-Slave Node	5, 6, 7, 26, 36 - 1, 3, 23, 33 4, 24

The contact input parameters are defined in the following table:

Input Data	Description
Static Friction Coefficient	Friction coefficient between the contact surfaces.
Penalty Scale Factor	Factor to scale forces between contact faces based on the penalty formulation (ie: forces proportional to the penetration depth).
Segment Thickness	The contact thickness indicates the distance away from a contact face where physical contact is established. PAMCRASH provides a default value (except for Type1) if none is specified.
Contact Search Acceleration	This represents the number of time steps between contact slave searches for contract types 5, 6, 7, 23, 24, and 26.
Stiffness Prop. Damping	Stiffness proportional damping ratio (value less than 1.0).
Activation Time	Activation time for this sliding interface.
Deactive Time	Deactivation time for this sliding interface. A value of 0 indicates that the interface remains active until end of run.

Note that there is a preview facility on the application Tool Form.

Geometric Rigid Wall

Object	Type	Analysis Type
Geometric Rigid Wall	Nodal	Structural

This LBC type is used to define a **RIGWA_/_** entry and an auxiliary **FUNCT_/_**entry if a motion time history is defined. The following table outlines the options:

Table 2-8 Geometric Rigid Wall Options

Object	Option 1	Option 2	Option 3
Geometric Rigid Wall	Prismatic Cylindrical Spherical	Static Defined Velocity Initial Velocity	Frictionless No Slip Frictional

The input data for geometric rigid walls are as follows:

Input Data	Description
Friction Coefficient	For frictional behavior only.
Mass	Mass of the rigid wall (assumed infinite if omitted). Applies for moving walls only.
Velocity <u,v,w>	Defines motion in the local coordinate system of the wall. It is used for infinite mass moving walls only. Note that this is a time dependent field. If this field has only one dependent variable this is assumed to be the velocity in the local z direction. To define other directions the field must have three dependent variables, representing the components of the velocity in the local x, y, z directions.
Centroid and Orientation	The local coordinate system used to define the entity. This must have the local z axis pointing outward from the wall. See manual for relationship to the geometry of the wall.
Edge Length (x) (y) (z)	Applies for Prism and flat surface.
Radius of Cylinder/Length of Cylinder	Applies for cylinder.
Radius of Sphere	Applies for sphere.

Note that you must select a local coordinate system that is used when generating the geometry of the wall (although the default global coordinate system can also be used). This coordinate system is centroidal-based not face-based as used by PAMCRASH. The Z-axis of this coordinate system defines the rigid wall outward normal. But note that the direction of motion is defined by the velocity vector not the outward normal vector. Note also that a facility for preview of the Rigid Wall and the slave nodes is provided on the input forms.

Planar Rigid Wall

Object	Type	Analysis Type
Planar Rigid Wall	Nodal	Structural

This LBC type is used to define a **RIGWA_/_** entry for an infinite rigid wall and an auxiliary **FUNCT_/_** entry if a motion time history is defined. The following table outlines the options:

Table 2-9 Planar Rigid Wall Options

Object	Option 1	Option 2
Planar Rigid Wall	Static Defined Velocity Initial Velocity	Frictionless No slip Frictional

The input data for planar rigid walls are as follows:

Input Data	Description
Friction Coefficient	For frictional behavior only.
Mass	Mass of the rigid wall (assumed infinite if omitted). Applies for moving walls only.
Velocity <u,v,w>	Defines motion in the local coordinate system of the wall. It is used for infinite mass moving walls only. Note that this is a time dependent field. If this field has only one dependent variable this is assumed to be the velocity in the local z direction. To define other directions the field must have three dependent variables, representing the components of the velocity in the local x, y, z directions.
Location and Orientation	The local coordinate system used to define the wall. This must have the local z axis pointing outward from the wall.

Note that you must select a local coordinate system that is used when generating the geometry of the wall (although the default global coordinate system can also be used). This coordinate system is centroidal-based not face-based as used by PAMCRASH. The Z-axis of this coordinate system defines the rigid wall outward normal. Note that the direction of motion is defined by the velocity vector, not the outward normal vector.

Nodal Rigid Body

Object	Type	Analysis Type
Rigid Body	Nodal	Structural

This LBC type is used to define a **RIGBO_/_** entry for a rigid body defined by an assembly of nodes. No input data is required for the Computed option. The following table outlines the input data for the Defined Locally option:

Input Data	Description
Mass	Translational mass of rigid body.
Inertia Ixx	xx component of inertia tensor.
Inertia Iyy	yy component of inertia tensor.
Inertia Izz	zz component of inertia tensor.
Local Coordinate Frame	Local coordinate system, used when defining centroid and inertia. The coordinate system must be placed at the required center of gravity. The required nodes will be generated automatically during translation.

Note that spotwelds and rivets are not supported.

Nodal Constraint

Object	Type	Analysis Type
Nodal Constraint	Nodal	Structural

This LBC type is used to define one or more **NODCO_/_** entries for a group of nodes. The only input data required for nodal constraints is Translations and Coordinate Frame. Note that rivets are not supported. Time history information is ignored. The scale factor has no effect.

Input Data	Description
Translations (T1,T2,T3)	Enter 0 for a translational constraint and “,” for translational freedom.

Nodal Damping

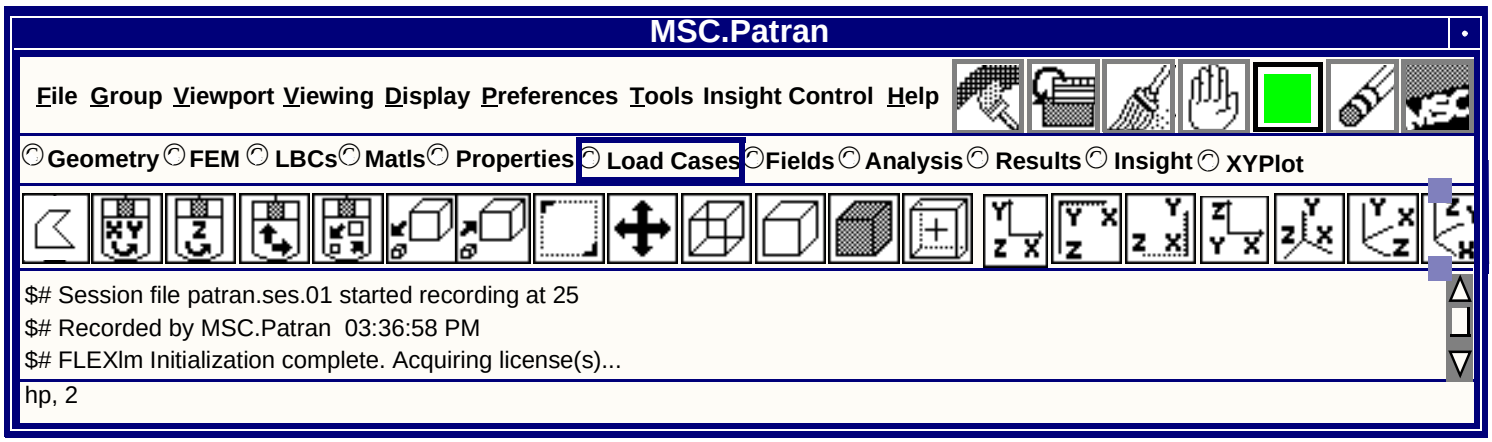
Object	Type	Analysis Type
Nodal Damping	Nodal	Structural

This LBC type is used to define a **DAMP__/_** entry for a group of nodes. Time history information is ignored. The scale factor has no effect.

Input Data	Description
Damping Factor q = c/m	Mass proportional nodal damping factor
Start Time	Starting time for damping
End Time	Removal time for damping

2.7 Load Cases

Load cases in MSC.Patran are used to group a series of load sets into one load environment for the model. Load cases are selected when preparing an analysis, not load sets. The usage for PAMCRASH is consistent, however only one loadcase can be selected for translation. For information on how to define static and / or transient load cases, see **Overview of the Load Cases Application** (Ch. 5) in the *MSC.Patran Reference Manual, Part 5: Functional Assignments.* Note that static load cases are not applicable to the PAMCRASH Preference and should not be used.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

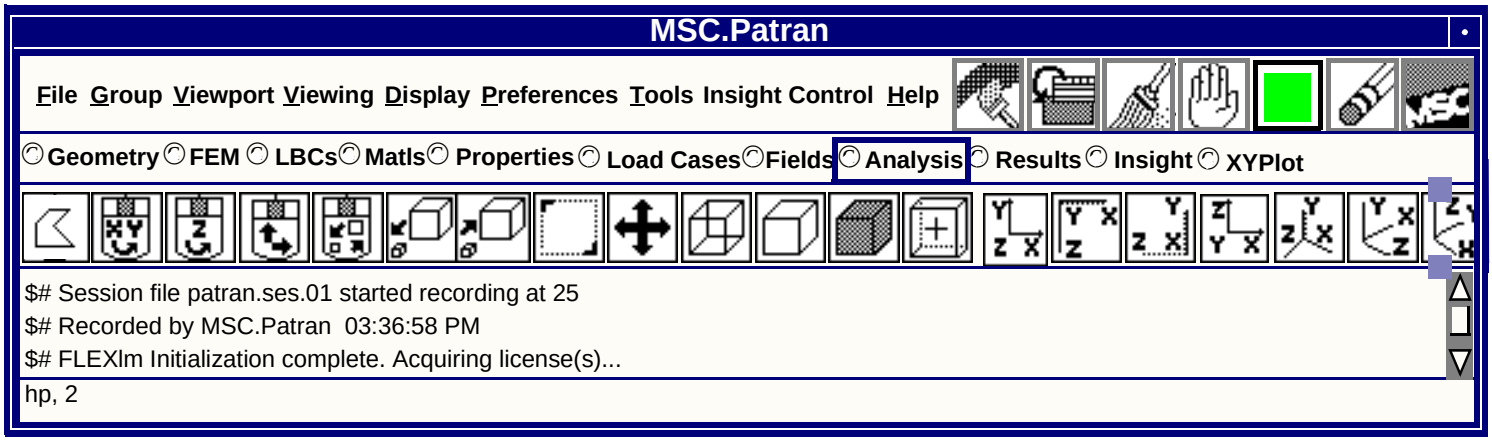
CHAPTER
3

Running an Analysis

- Review of the Analysis Form
- Translation Parameters
- Solution Parameters
- Select Load Case
- Output Requests
- Output Controls
- Select Group
- Setting PAMCRASH IDs

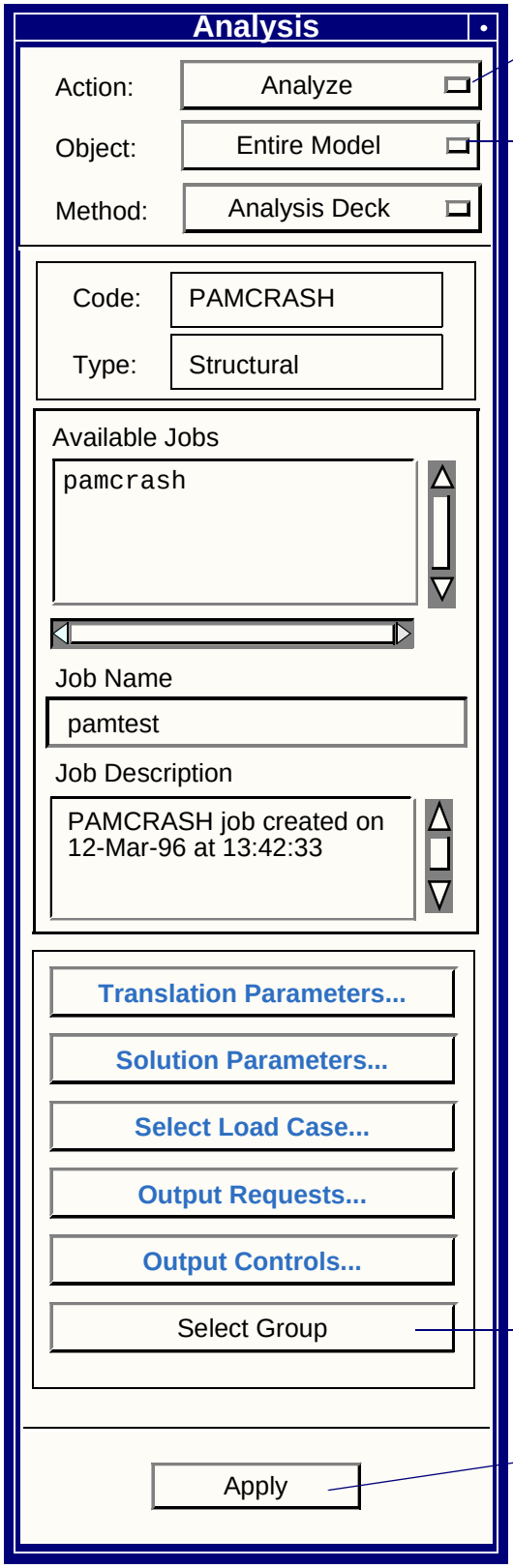
3.1 Review of the Analysis Form

The Analysis form appears when the Analysis toggle, located on the MSC.Patran main form is chosen. To run an analysis, or to create a PAMCRASH input file, select Analyze as the Action on the Analysis form. Other forms brought up by the Analysis form are used to define and control the analysis to be conducted and to set global defaults, where appropriate. These forms are described on the following pages. For further information see **The Analysis Form** (p. 8) in the *MSC.Patran Reference Manual, Part 5: Analysis Application*.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Analysis Form

This form appears when the Analysis toggle is chosen on the main form. When preparing for an analysis run, select Analyze as the Action.



Analysis Options for Action are:
Analyze, Read Input File, and Delete.

Options for Object depend upon the Action selected. For “Analyze” these are:

- Entire Model
- Select Group

These selections apply only when the action is “Analyze” and the object is “Entire Model”.

Appears only when the Object is set to Select Group.

Check for error and warning messages on the terminal after translation.

The Object indicates which part of the model is to be analyzed.

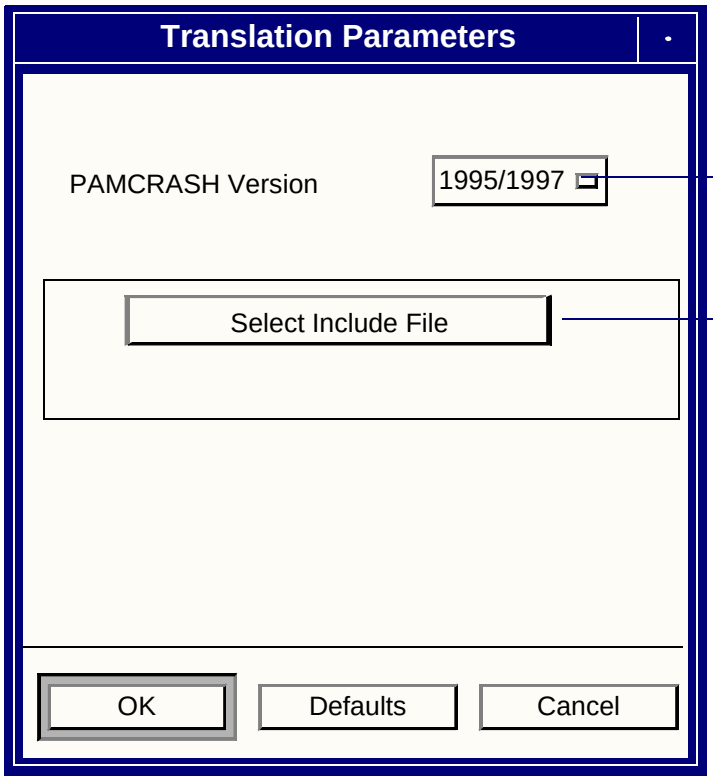
- **Entire Model** is selected if the whole model is to be analyzed.
- **Select Group** allows one or more groups to be selected from a form and written to the deck.

The Method indicates how far the translation is to be taken.

- **Analysis Deck** is selected if an analysis file translation is to be done, plus all load case, analysis type and analysis parameter data are to be translated. A complete input file, ready for PAMCRASH, should be generated.

3.2 Translation Parameters

The Translation Parameters form allows the user to control the manner in which the PAMCRASH input file is generated.

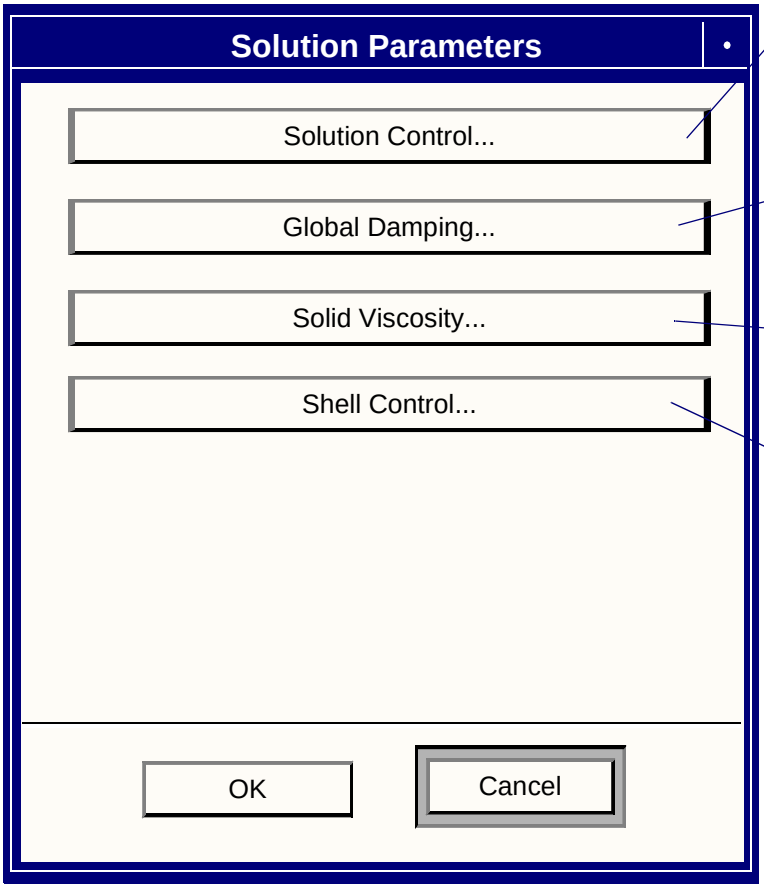


At present only the 1995/1997 version of PAMCRASH is supported.

The **"INCLU_/_"** control command allows the inclusion of files containing any subset of the input data. Choosing "Select Include File" allows the user to identify a single file, the name of which is written on the **"INCLU_/_"** entry.

3.3 Solution Parameters

The Solution Parameters form provides access to subordinate forms upon which are defined the parameters controlling execution of a PAMCRASH analysis.



Define parameters controlling execution, including CPU limit and timestep control. Data is written to the **CPULIMIT, CTRL_ _/_, MNTR_ _/_** and **DATACHECK** entries.

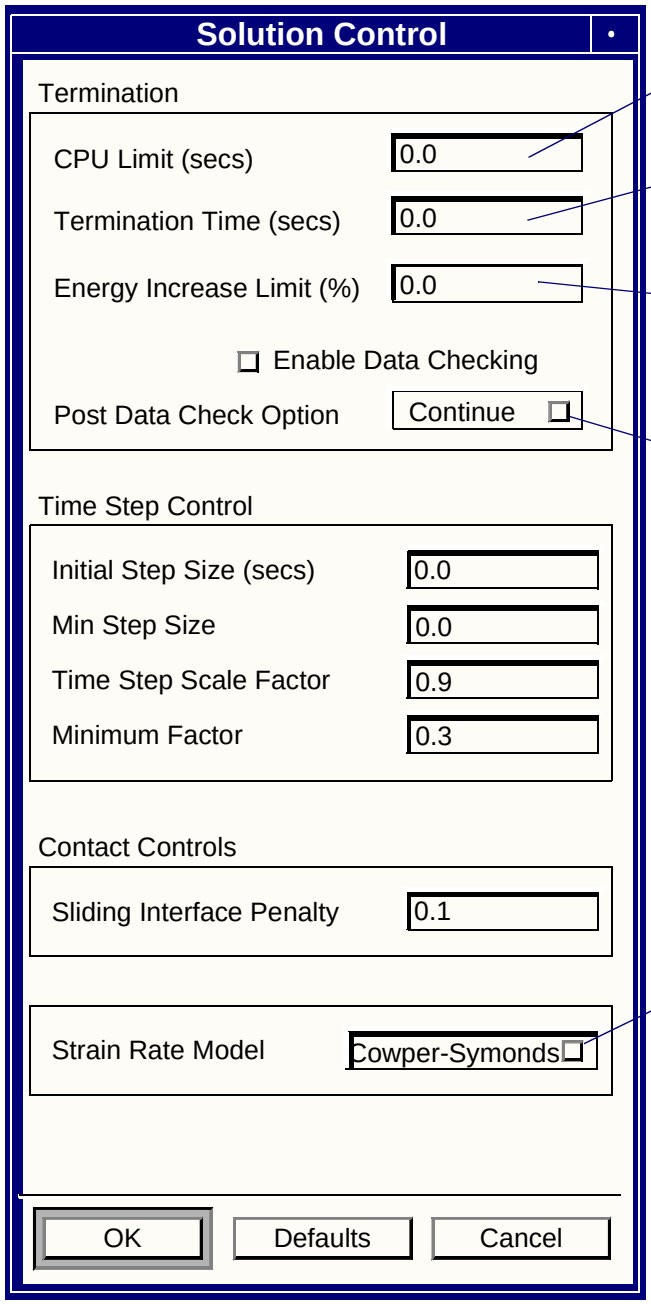
Global Damping parameters for the **CTRL_ _/ _**entry are defined via the “Global Damping” form.

Solid viscosity parameters for the **CTRL_ _/_** entry are defined via the “Solid Viscosity” form.

Shell Control parameters for the **SHELLCHECK and CTRL_ _/_**entries are defined via the “Shell Control” form.

Solution Control

The Solution Control subordinate form defines data to be written to the **CPULIMIT, CTRL_ _/_, MNTR_ _/_** and **DATACHECK** entries.



CPU time limit for job termination in seconds.

Analysis time limit for job termination.

% change in energy increase for termination of the calculation. The default, 0.0, renders this inactive.

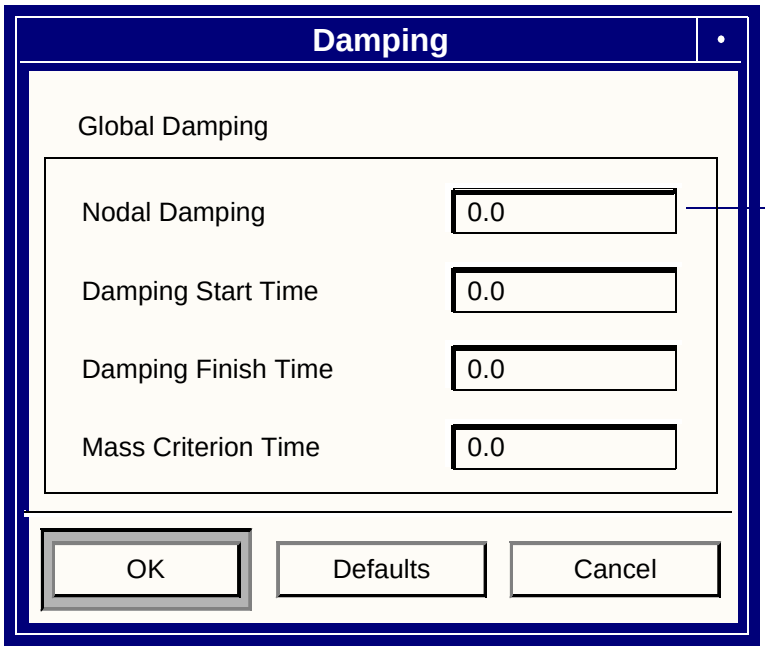
Options for Post Data Check are as follows:

1. Continue
2. Stop (Restart)
3. Stop (No Restart)

Cowper-Symonds
Johnson-Cook
Modified Jones
Left Shifted
Krupkowsky

Global Damping

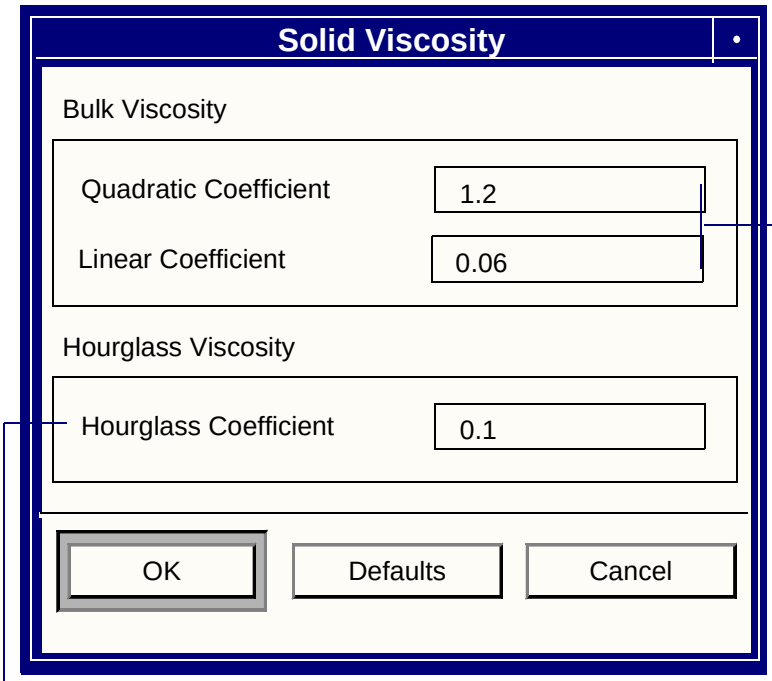
The Global Damping subordinate form defines data to be written to the **CTRL__/_** entry.



Mass proportional nodal damping factor (q=c/m).

Solid Viscosity

The Solid Viscosity subordinate form defines data to be written to the **CTRL_ _/_** entry.

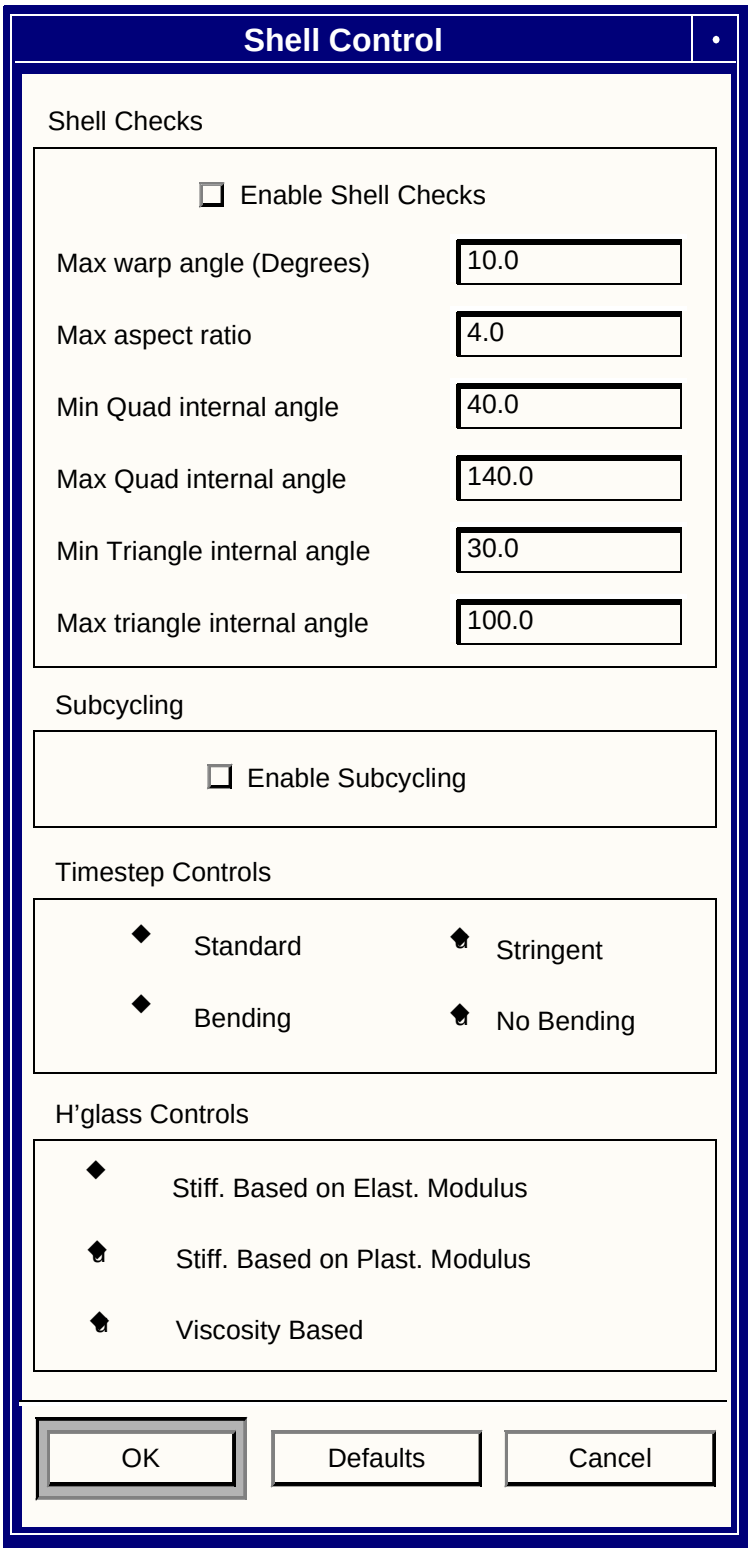


These define the values of the bulk viscosity coefficients for solid elements.

This defines the hourglass viscosity coefficient for solid elements. Values between 0.05 and 0.15 are recommended.

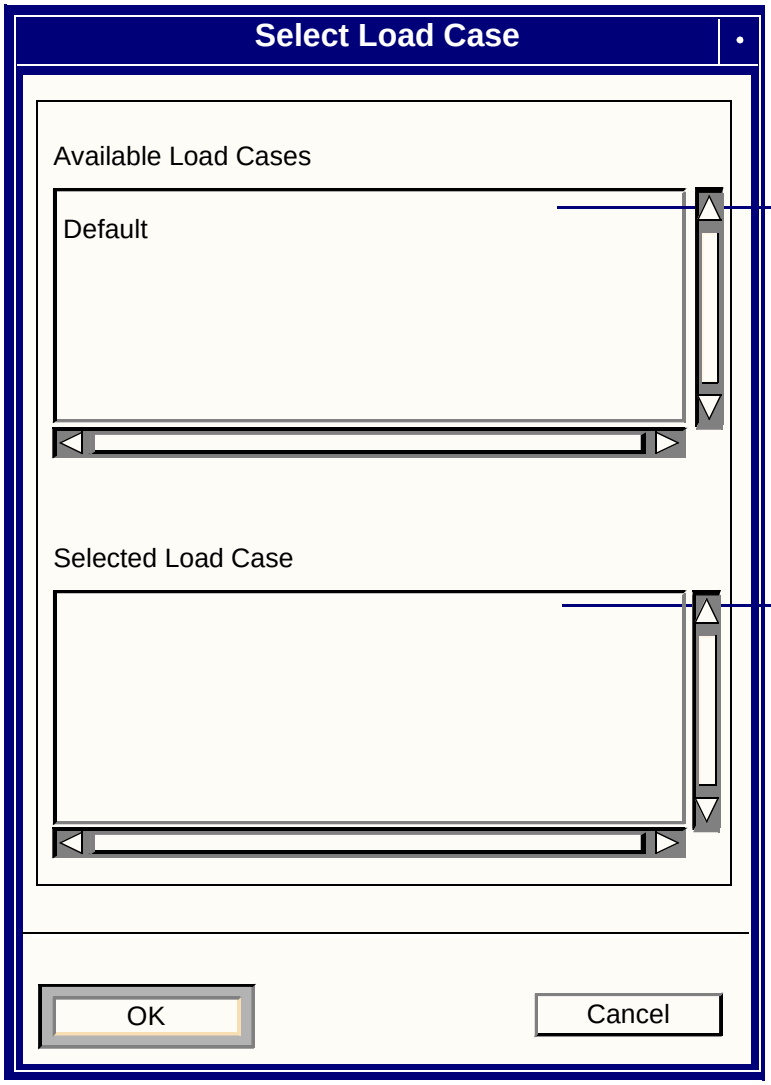
Shell Control

The Shell Control subordinate form defines data to be written to the **SHELLCHECK** entry.



3.4 Select Load Case

This form appears when the Select Load Case button is selected on the Analysis form. Use this form to select the load case to be included in the job.

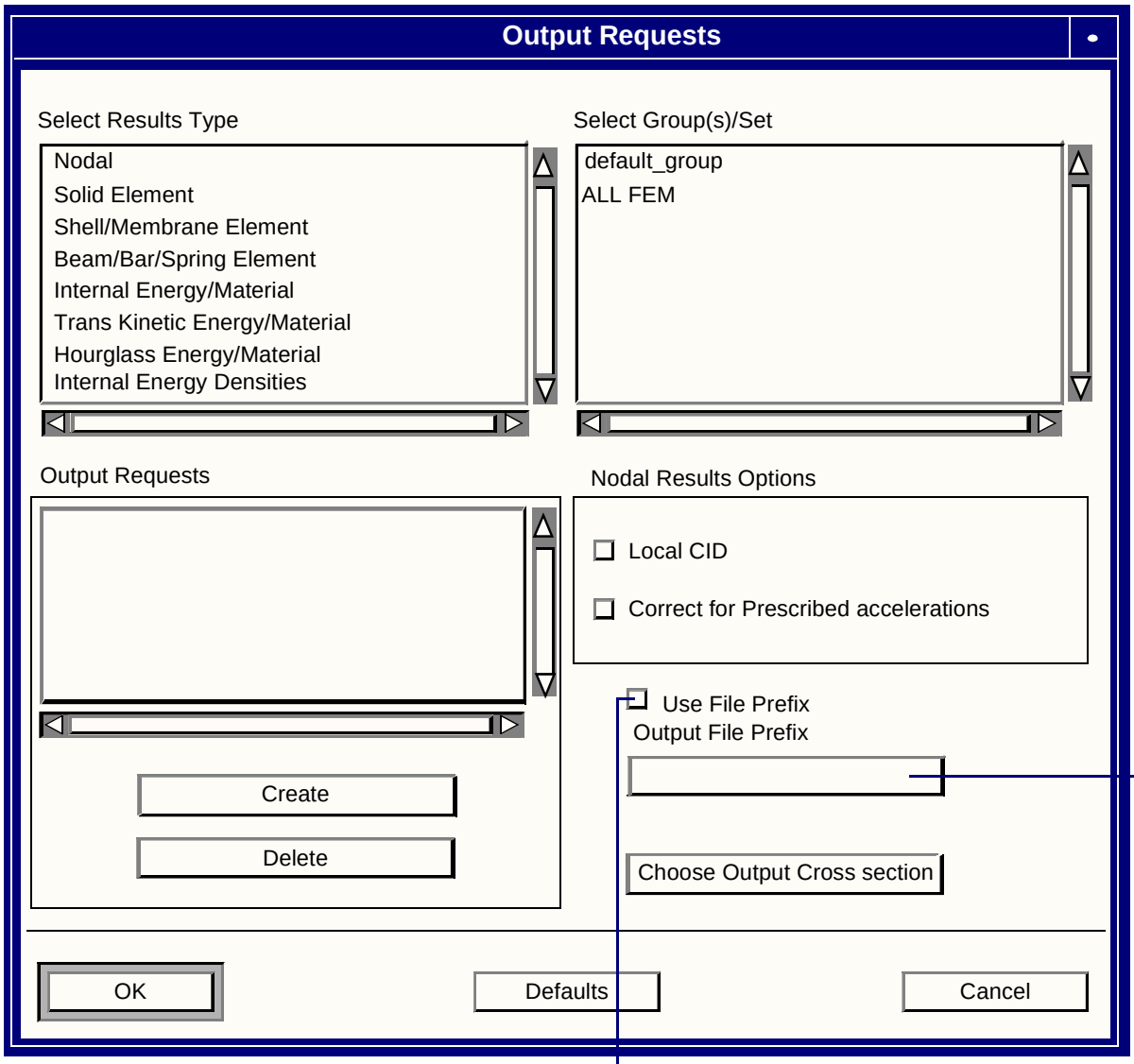


Displays the list of all load cases currently in the database. The desired load cases may be selected from this area.

Only one load case can be selected. The default is the current loadcase.

3.5 Output Requests

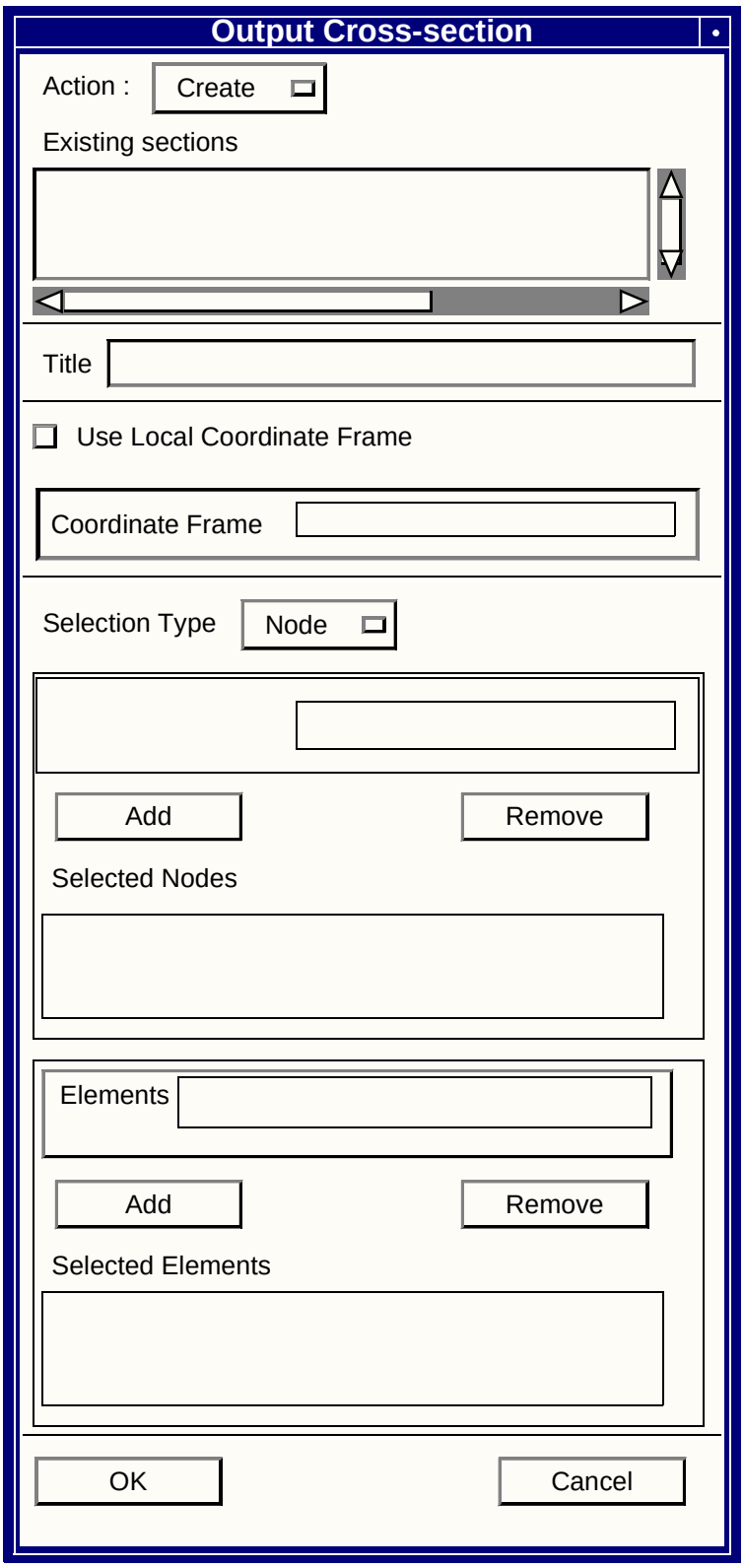
The Output Requests form allows definition of what data is desired from the analysis code in the form of results. The settings can be accepted, as altered, by selecting the OK button on the bottom of the form. If the Cancel button is selected, the form will be closed without any of the changes being accepted. Selecting the Defaults button resets the form to the initial default settings.



y default the toggle is ON. If the toggle is switched
FF, the Output File Prefix databox is then disabled.
n this situation, the FILE card is not written to the
amcrash deck, and, as before, the analysis
obname is used for the file prefix of the Pamcrash

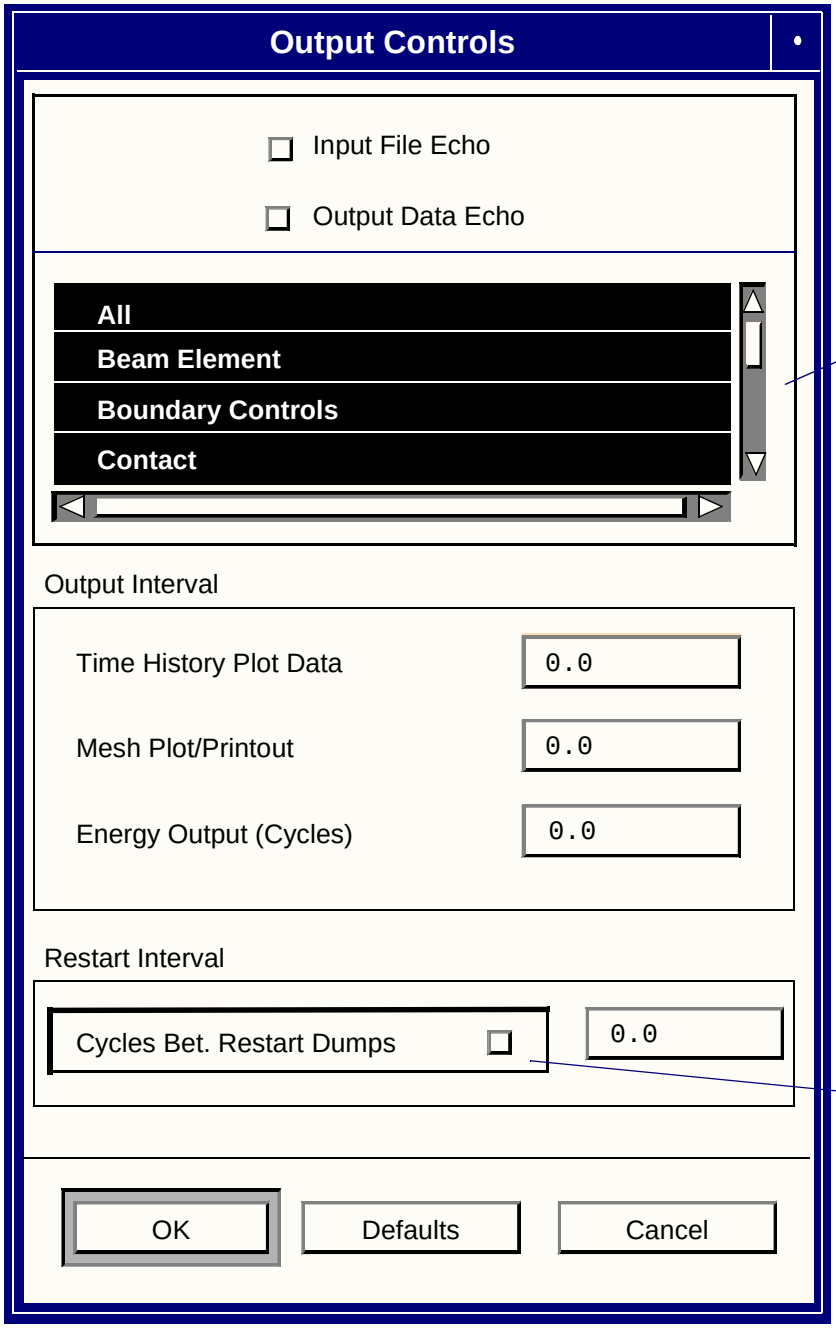
This can be used to define the prefix of the Pamcrash deck. It is also used to define the FILE card in the Pamcrash deck. The FILE card is used to define the prefix of all output files used by the Pamcrash solver.

This form appears when the Choose Output Cross Section button is selected on the Output Requests form. The Output Cross-section form defines data to be written to the TRAFO_/_ and SECFO_/_ entries.



3.6 Output Controls

The Output Controls form provides control over data generated during execution. This data is entered on the **LIST_ _/_, PRINT_ _/_** and **CTRL_ _/_** entries. The settings can be accepted, as altered, by selecting the *OK button* on the bottom of the form. If the *Cancel button* is selected instead, the form will be closed without any of the changes being accepted. Selecting the *Defaults button* resets the form to the default settings.



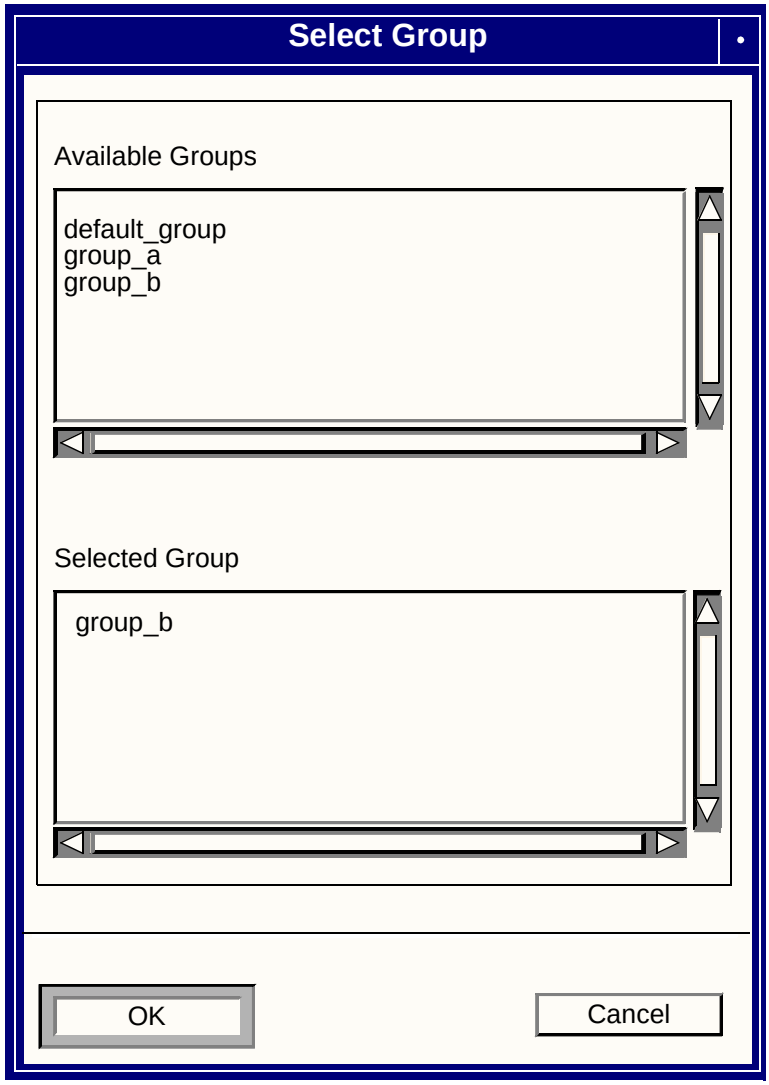
These selections control the data sets written to the output file.

Options are:

Cycles Bet. Restart Dumps and Internal Bet. Restart Dumps.

3.7 Select Group

The Select Group form allows the user to select any of the groups in the model and write them to the deck.



3.8 Setting PAMCRASH IDs

Normally the Pamcrash keyword ID is set using the corresponding PATRAN entity ID. However the user can set IDs by naming the Patran entites NAME.### where ### is the required ID.

For the Pamcrash writer, only the Patran property set ID is used in the deck. The material ID used in the deck comes from the property set. Therefore, if the user requires a material of ID 505, the property set that references the material can be named 'Pset.505'.

Note: For multiple Psets, the user must assign each Pset a unique .### for the suffix (not zero) otherwise the code that sets IDs from names will not be called.

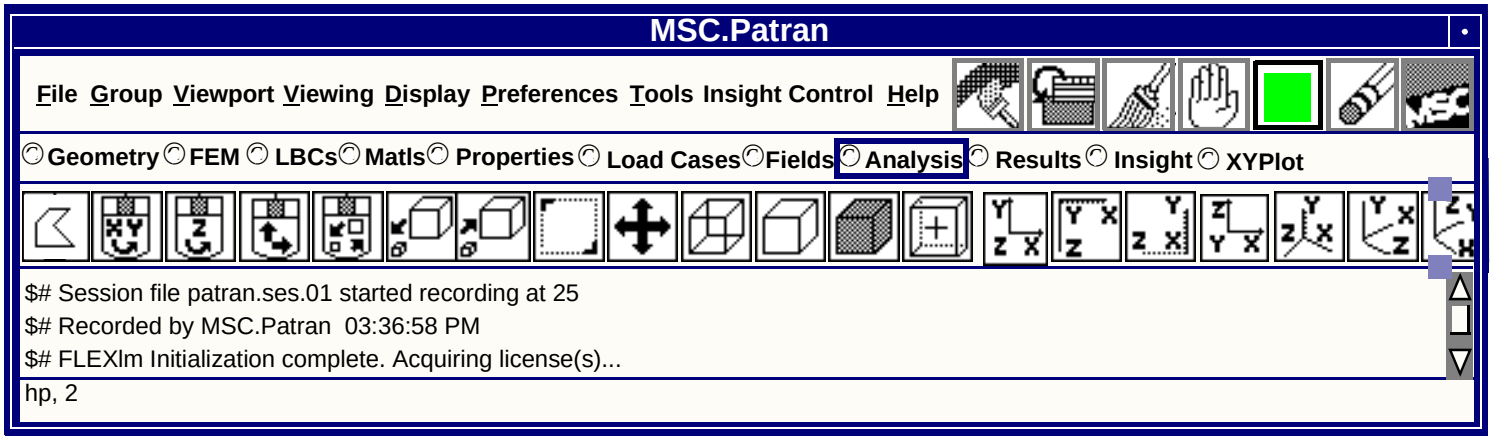
CHAPTER

4 Read Input File

- Review of Read Input File Form
- Selection of Input File
- Data Translated from the PAMCRASH Input File
- Reject File

4.1 Review of Read Input File Form

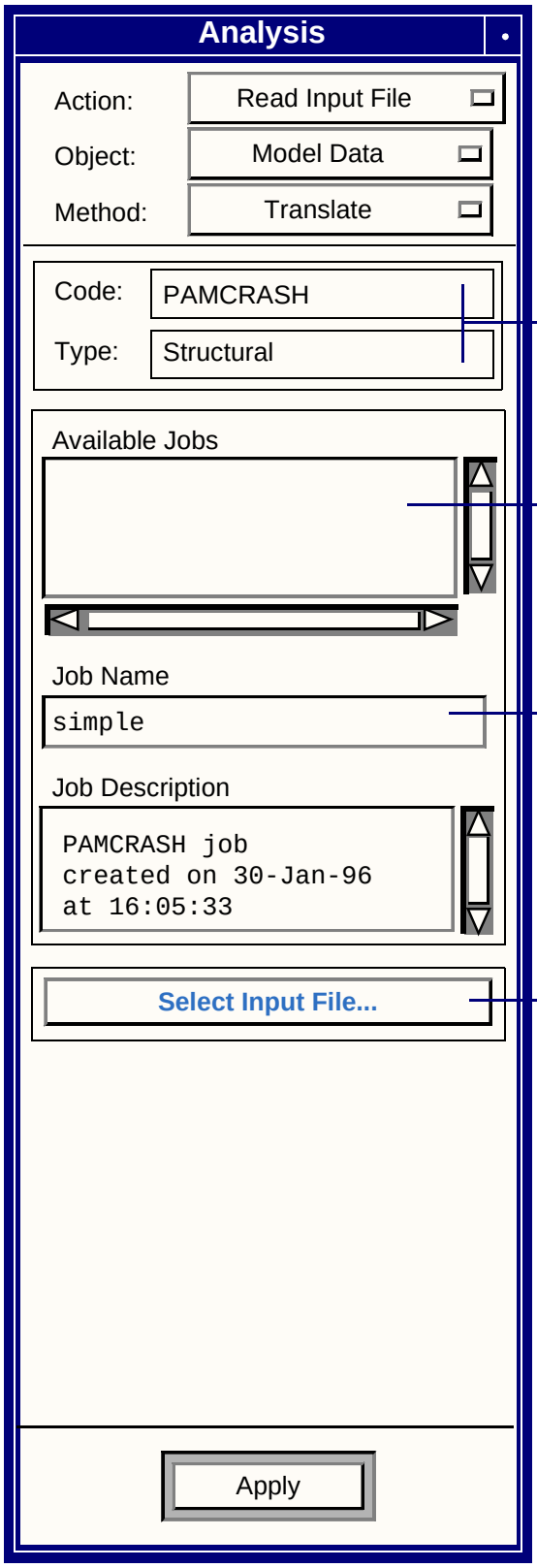
The Analysis form will appear when the Analysis toggle, located on the MSC.Patran main form, is chosen.


MSC.Patran
File Group Viewport Viewing Display Preferences Tools Insight Control Help
Geometry FEM LBCs Matls Properties Load Cases Fields Analysis Results Insight XYPlot
$# Session file patran.ses.01 started recording at 25
$# Recorded by MSC.Patran 03:36:58 PM
$# FLEXlm Initialization complete. Acquiring license(s)...
hp, 2

Read Input File as the selected Action on the Analysis form allows some of the model data from PAMCRASH input file to be translated into the MSC.Patran database. A subordinate Select Input File form allows the user to specify the PAMCRASH input file to translate.

Read Input File Form

This form appears when the Analysis toggle is selected on the main form. Read Input File, as the selected Action, specifies that model data is to be translated from the specified PAMCRASH input file into the MSC.Patran database.



Indicates the selected Analysis Code and Analysis Type, as defined in the **Preferences>Analysis** (p. 343) in the *MSC.Patran Reference Manual, Part 1: Basic Functions*.

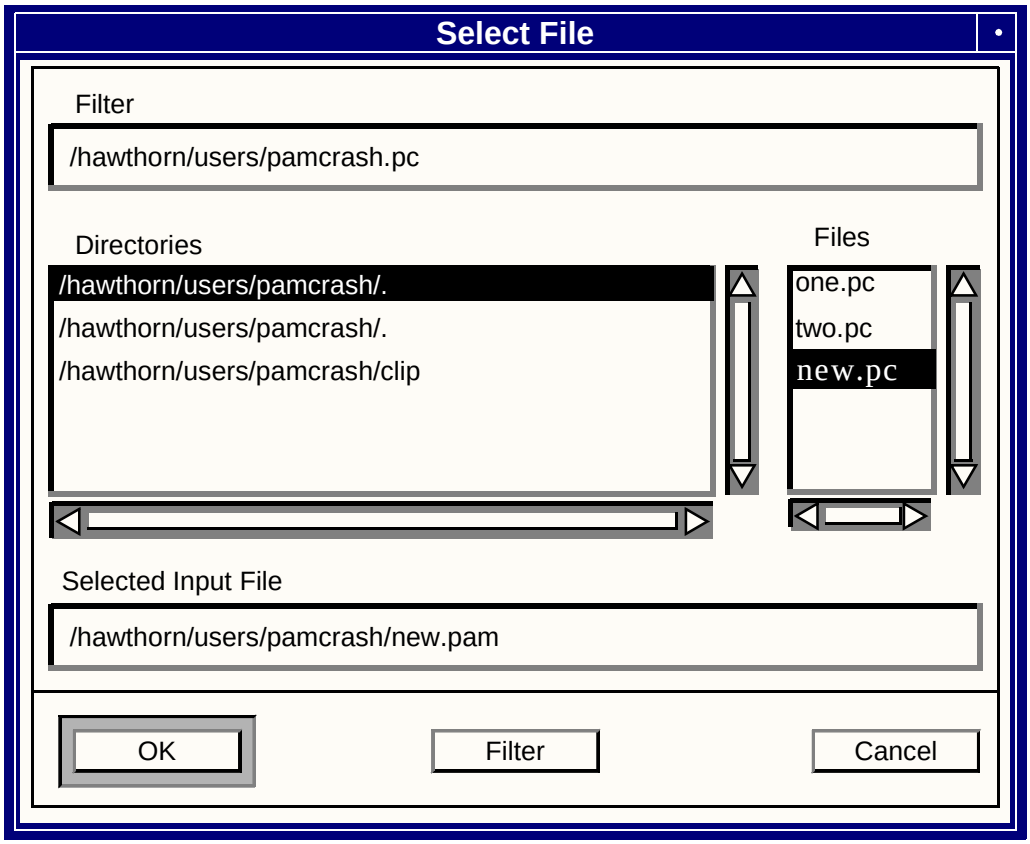
List of already existing jobs.

Name assigned to current translation job. This job name will be used as the base file name for the message file.

Activates a subordinate Select Input File form which allows the user to specify the PAMCRASH input file to be translated.

4.2 Selection of Input File

This subordinate form appears when the Select Input File button is selected on the Analysis form when Read Input File is the selected Action. It allows the user to specify which PAMCRASH input file to translate.



4.3 Data Translated from the PAMCRASH Input File

The following is a list of the data supported.

Table 4-1 Input File Translation Data

File Section	Keyword
Control	NOLIST / LIST (Listing Control)
	NOPRINT / PRINT (Printing Control)
	MNTR (Monitoring)
	SHELLCHECK (Shell Geometry Limits)
	DATACHECK (Date Checking)
	TIMESTEP (Shell Time Step Control)
Nodes	FRAME_ / _ (Coordinate Frame)
	NODE_ _ / _ (Nodal Point Data)
	MASS_ _ / _ (Added Mass)
Elements	SOLID _ / _(Solid Elements)
	SHELL_ / _(Shell Elements)
	MEMBR_ / _(Membrane Elements)
	BEAM_ / _(Beam Elements)
	BAR_ _ _ / _(Bar Elements)
	SPRING / _(Spring Elements)
	JOINT_ / _(Joint Elements)
	MASS_ / _(Mass Elements)
Materials	MAT_ / _(Materials)
Ply Data	Ply_ _ _ / _(Composite)
Plot Output	THLNO_ / _(Nodal Time History)
	THLSO_ / _(Solid Element Output)
	THLSH_ / _(Shell and Membrane Element Output)
	THLBM_ / _(Beam,Bar,Spring, Deshpot etc Output)
	TRAFO_ / _(Cross Section For Force Output)
	SECFO_ / _(Cross Section For Force Output)

Table 4-1 Input File Translation Data

File Section	Keyword
LBCs	BOUNC_/_(Displacement)
	CONLO_/_(Forces)
	DAMP_/_(Nodal Damping)
	INVEL_/_(Initial Velocity)
	NODCO_/_(Nodal Constraint)
	RIGBO_/_(Nodal Rigid Body)
	RIGWA_/_(Rigid Wall) Planar or Geometric
	SLINT_/_(Contact)
	SLINT2_/_(Contact)
	VELBC_/_(Velocity)

4.4 Reject File

The input file reader places all unsupported Pamcrash keywords in a reject file which has the extension .rej

CHAPTER 5

Files

- Files

5.1 Files

The MSC.Patran PAMCRASH Preference uses or creates several files.The following table outlines each file and its uses. In the file name definition, **jobname** will be replaced with the jobname assigned by the user.

File Name	Description
***.db**	This is the MSC.Patran database. During an analyze pass, model data is read from this database. This file typically resides in the current directory.
jobname.pc	This is the PAMCRASH input file created by the interface. This file typically resides in the current directory.
pat3pam -j	This is the actual forward translation program, translating between the MSC.Patran database and a PAMCRASH input file. It is typically run within MSC.Patran, transparent to the user, but can also be run independently. For example: **<installation_directory>/bin/exe/pat3pam -j my_job -d my_database.db > my_job.msg &** MSC.Patran searches its file path for this file, but it typically resides in the <installation_directory> / bin / exe directory.
pat3pam -i	This is the PAMCRASH input file reader program. It is typically run within MSC.Patran, transparent to the user, but can also be run independently with the following command; <installation_directory> / bin / exe / pat3pam -i my_deck.pc -d my_database.db > my_job.msg & MSC.Patran searches its file path for this file, but it typically resides in the `<installation_directory>/bin/exe` directory.

I N D E X

B

bulk data file, 74

C

coordinate frames, 12

D

databases
  MSC.Patran template, 4

E

elastoplastic, 20
element properties, 27
elements
  scalar spring, 33
  solid, 36
  standard homogeneous plate, 34

F

files, 82
finite elements, 13, 15

I

input file, 74

L

load cases, 55
loads and boundary conditions, 37

M

materials, 16

N

nodes, 14

P

preferences, 8
properties, 27

R

read input file, 74

S

supported entities, 9

T

template database, 4